Report #1082

LS-DYNA3D
USER'S MANUAL

(Nonlinear Dynamic Analysis of Structures in Three Dimensions)

August 1, 1995
Version 936

copyright 1992-1995
all rights reserved

LIVERMORE SOFTWARE
TECHNOLOGY CORPORATION

Mailing Address:

Livermore Software Technology Corporation
2876 Waverley Way
Livermore, Ca 94550

FAX: 510-449-2507

TEL: 510-449-2500

Copyright 1995, LSTC
All rights reserved

TABLE OF CONTENTS

ABSTRACT **I.1**

INTRODUCTION **I.3**

CHRONOLOGICAL HISTORY I.3

DESCRIPTION OF KEYWORD INPUT I.8

MATERIAL MODELS I.20

SPATIAL DISCRETIZATION I.22

SLIDING INTERFACES I.25

INTERFACE DEFINITIONS FOR COMPONENT ANALYSIS I.27

CAPACITY I.29

CODE ORGANIZATION I.30

SENSE SWITCH CONTROLS I.31

PRECISION I.32

EXECUTION SYNTAX I.33

RESTART ANALYSIS I.37

VDA/IGES DATABASES I.39

MESH GENERATION I.40

LS-TAURUS POST-PROCESSING I.41

EXECUTION SPEEDS I.43

UNITS I.45

GENERAL CARD FORMAT I.46

MPP/LS-DYNA3D USER INFORMATION I.47

***AIRBAG** **1.1**

*AIRBAG_OPTION 1.1

*AIRBAG_INTERACTION 1.19

*AIRBAG_REFERENCE_GEOMETRY 1.20

***ALE** **2.1**

*ALE_SMOOTHING 2.1

***BOUNDARY** **3.1**

*BOUNDARY_CONVECTION_OPTION 3.2

*BOUNDARY_CYCLIC 3.4

*BOUNDARY_FLUX_OPTION 3.6

*BOUNDARY_NON_REFLECTING 3.9

TABLE OF CONTENTS

*BOUNDARY_PRESCRIBED_MOTION_OPTION 3.10
*BOUNDARY_PRESSURE_OUTFLOW_OPTION 3.12
*BOUNDARY_RADIATION_OPTION 3.14
*BOUNDARY_SLIDING_PLANE 3.16
*BOUNDARY_SPC_OPTION 3.17
*BOUNDARY_SYMMETRY_FAILURE 3.18
*BOUNDARY_TEMPERATURE_OPTION 3.19
*BOUNDARY_USA_SURFACE 3.20

***CONSTRAINED 4.1**
*CONSTRAINED_EXTRA_NODES_OPTION 4.2
*CONSTRAINED_GENERALIZED_WELD_OPTION 4.3
*CONSTRAINED_JOINT_OPTION 4.10
*CONSTRAINED_JOINT_STIFFNESS_OPTION 4.13
*CONSTRAINED_LINEAR 4.22
*CONSTRAINED_NODAL_RIGID_BODY_{OPTION} 4.25
*CONSTRAINED_NODE_SET 4.29
*CONSTRAINED_RIGID_BODIES 4.31
*CONSTRAINED_RIGID_BODY_STOPPERS 4.32
*CONSTRAINED_RIVET 4.35
*CONSTRAINED_SHELL_IN_SOLID 4.36
*CONSTRAINED_SHELL_TO_SOLID 4.37
*CONSTRAINED_SPOTWELD 4.39
*CONSTRAINED_TIE-BREAK 4.41
*CONSTRAINED_TIED_NODES_FAILURE 4.42

***CONTACT 5.1**
*CONTACT_{OPTION1}_{OPTION2}_{OPTION3} 5.1
*CONTACT_ENTITY 5.19
*CONTACT_1D 5.27

***CONTROL 6.1**
*CONTROL_ADAPTIVE 6.2
*CONTROL_ALE 6.4
*CONTROL_BULK_VISCOSITY 6.6
*CONTROL_CONTACT 6.7
*CONTROL_COUPLING 6.11
*CONTROL_CPU 6.13

*CONTROL_DYNAMIC_RELAXATION 6.14
*CONTROL_ENERGY 6.16
*CONTROL_HOURGLASS 6.17
*CONTROL_OUTPUT 6.18
*CONTROL_PARALLEL 6.19
*CONTROL_SHELL 6.20
*CONTROL_SOLUTION 6.22
*CONTROL_STRUCTURED 6.23
*CONTROL_SUBCYCLE 6.24
*CONTROL_TERMINATION 6.25
*CONTROL_THERMAL_NONLINEAR 6.26
*CONTROL_THERMAL_SOLVER 6.27
*CONTROL_THERMAL_TIMESTEP 6.29
*CONTROL_TIMESTEP 6.30

***DAMPING 7.1**
*DAMPING_GLOBAL 7.1
*DAMPING_PART_MASS 7.3
*DAMPING_PART_STIFFNESS 7.4

***DATABASE 8.1**
*DATABASE_OPTION 8.2
*DATABASE_BINARY_OPTION 8.4
*DATABASE_CROSS_SECTION_OPTION 8.6
*DATABASE_EXTENT_OPTION 8.9
*DATABASE_HISTORY_OPTION 8.16
*DATABASE_NODAL_FORCE_GROUP 8.17
*DATABASE_SPRING_FORWARD 8.18
*DATABASE_SUPERPLASTIC_FORMING 8.19
*DATABASE_TRACER 8.20

***DEFINE 9.1**
*DEFINE_BOX 9.2
*DEFINE_COORDINATE_NODES 9.3
*DEFINE_COORDINATE_SYSTEM 9.4
*DEFINE_COORDINATE_VECTOR 9.6
*DEFINE_CURVE 9.7
*DEFINE_SD_ORIENTATION 9.9

TABLE OF CONTENTS

*DEFINE_TABLE 9.10
*DEFINE_VECTOR 9.12

***DEFORMABLE**_TO_RIGID **10.1**
*DEFORMABLE_TO_RIGID 10.2
*DEFORMABLE_TO_RIGID_AUTOMATIC 10.3
*DEFORMABLE_TO_RIGID_INERTIA 10.7

***ELEMENT** **11.1**
*ELEMENT_BEAM_OPTION 11.2
*ELEMENT_DISCRETE 11.5
*ELEMENT_MASS 11.6
*ELEMENT_SEATBELT 11.7
*ELEMENT_SEATBELT_ACCELEROMETER 11.8
*ELEMENT_SEATBELT_PRETENSIONER 11.9
*ELEMENT_SEATBELT_RETRACTOR 11.11
*ELEMENT_SEATBELT_SENSOR 11.17
*ELEMENT_SEATBELT_SLIPRING 11.21
*ELEMENT_SHELL_OPTION 11.23
*ELEMENT_SOLID_OPTION 11.28
*ELEMENT_TSHELL 11.33

***EOS** **12.1**
*EOS_LINEAR_POLYNOMIAL 12.2
*EOS_JWL 12.4
*EOS_SACK_TUESDAY 12.5
*EOS_GRUNEISEN 12.6
*EOS_RATIO_OF_POLYNOMIALS 12.8
*EOS_LINEAR_POLYNOMIAL_WITH_ENERGY_LEAK 12.12
*EOS_IGNITION_AND_GROWTH_OF_REACTION_IN_HE 12.13
*EOS_TABULATED_COMPACTION 12.16
*EOS_TABULATED 12.19
*EOS_PROPELLANT_DEFLAGRATION 12.21
*EOS_TENSOR_PORE_COLLAPSE 12.26

***HOURGLASS** **13.1**
*HOURGLASS 13.1

TABLE OF CONTENTS

***INCLUDE 14.1**

*INCLUDE 14.1

***INITIAL 15.1**

*INITIAL_DETONATION 15.2

*INITIAL_MOMENTUM 15.4

*INITIAL_STRESS_BEAM 15.5

*INITIAL_STRESS_SHELL 15.7

*INITIAL_STRESS_SOLID 15.9

*INITIAL_TEMPERATURE_OPTION 15.11

*INITIAL_VELOCITY 15.12

*INITIAL_VELOCITY_NODE 15.14

*INITIAL_VELOCITY_GENERATION 15.15

***INTEGRATION 16.1**

*INTEGRATION_BEAM 16.1

*INTEGRATION_SHELL 16.6

***INTERFACE 17.1**

*INTERFACE_COMPONENT_OPTION 17.1

*INTERFACE_LINKING_DISCRETE_NODE_OPTION 17.2

*INTERFACE_LINKING_SEGMENT 17.3

*INTERFACE_LINKING_EDGE 17.4

*INTERFACE_JOY 17.5

*INTERFACE_SPRINGBACK 17.6

***LOAD 18.1**

*LOAD_BEAM_OPTION 18.2

*LOAD_BODY_OPTION 18.4

*LOAD_BODY_GENERALIZED 18.6

*LOAD_BRODE 18.8

*LOAD_DENSITY_DEPTH 18.10

*LOAD_HEAT_GENERATION_OPTION 18.11

*LOAD_NODE_OPTION 18.12

*LOAD_RIGID_BODY 18.14

*LOAD_SEGMENT 18.16

*LOAD_SEGMENT_SET 18.17

*LOAD_SHELL_OPTION 18.19

TABLE OF CONTENTS

*LOAD_SUPERPLASTIC_FORMING 18.20
*LOAD_THERMAL_OPTION 18.23
*LOAD_THERMAL_CONSTANT 18.24
*LOAD_THERMAL_CONSTANT_NODE 18.25
*LOAD_THERMAL_LOAD_CURVE 18.26
*LOAD_THERMAL_TOPAZ 18.27
*LOAD_THERMAL_VARIABLE 18.28
*LOAD_THERMAL_VARIABLE_NODE 18.30

***MAT 19.1**
*MAT_ELASTIC_OPTION 19.4
*MAT_OPTION TROPIC_ELASTIC 19.7
*MAT_PLASTIC_KINEMATIC 19.13
*MAT_ELASTIC_PLASTIC_THERMAL 19.16
*MAT_SOIL_AND_FOAM 19.19
*MAT_VISCOELASTIC 19.23
*MAT_BLATZ-KO_RUBBER 19.24
*MAT_HIGH_EXPLOSIVE_BURN 19.25
*MAT_NULL 19.27
*MAT_ELASTIC_PLASTIC_HYDRO 19.29
*MAT_STEINBERG 19.32
*MAT_ISOTROPIC_ELASTIC_PLASTIC 19.36
*MAT_ISOTROPIC_ELASTIC_FAILURE 19.37
*MAT_SOIL_AND_FOAM_FAILURE 19.39
*MAT_JOHNSON_COOK 19.40
*MAT_PSEUDO_TENSOR 19.44
*MAT_ORIENTED_CRACK 19.49
*MAT_POWER_LAW_PLASTICITY 19.50
*MAT_STRAIN_RATE_DEPENDENT_PLASTICITY 19.52
*MAT_RIGID 19.55
*MAT_ORTHOTROPIC_THERMAL 19.59
*MAT_COMPOSITE_DAMAGE 19.62
*MAT_TEMPERATURE_DEPENDENT_ORTHOTROPIC 19.65
*MAT_PIECEWISE_LINEAR_PLASTICITY 19.68
*MAT_GEOLOGIC_CAP_MODEL 19.72
*MAT_HONEYCOMB 19.79
*MAT_MOONEY-RIVLIN_RUBBER 19.86

*MAT_RESULTANT_PLASTICITY 19.89
*MAT_FORCE_LIMITED 19.90
*MAT_CLOSED_FORM_SHELL_PLASTICITY 19.96
*MAT_FRAZER_NASH_RUBBER_MODEL 19.97
*MAT_LAMINATED_GLASS 19.100
*MAT_BARLAT_ANISOTROPIC_PLASTICITY 19.102
*MAT_FABRIC 19.105
*MAT_PLASTIC_GREEN-NAGHDI_RATE 19.109
*MAT_3-PARAMETER_BARLAT 19.110
*MAT_TRANSVERSELY_ANISOTROPIC_ELASTIC_PLASTIC 19.114
*MAT_BLATZ-KO_FOAM 19.117
*MAT_FLD_TRANSVERSELY_ANISOTROPIC 19.119
*MAT_NONLINEAR_ORTHOTROPIC 19.121
*MAT_USER_DEFINED_MATERIAL_MODELS 19.125
*MAT_BAMMAN 19.128
*MAT_BAMMAN_DAMAGE 19.134
*MAT_CLOSED_CELL_FOAM 19.137
*MAT_ENHANCED_COMPOSITE_DAMAGE 19.140
*MAT_LOW_DENSITY_FOAM 19.145
*MAT_COMPOSITE_FAILURE_MODEL 19.149
*MAT_ELASTIC_WITH_VISCOSITY 19.153
*MAT_KELVIN-MAXWELL_VISCOELASTIC 19.157
*MAT_VISCOUS_FOAM 19.159
*MAT_CRUSHABLE_FOAM 19. 161
*MAT_RATE_SENSITIVE_POWERLAW_PLASTICITY 19.163
*MAT_MODIFIED_ZERILLI_ARMSTRONG 19.165
*MAT_LINEAR_ELASTIC_DISCRETE_BEAM 19.168
*MAT_NONLINEAR_ELASTIC_DISCRETE_BEAM 19.170
*MAT_NONLINEAR_PLASTIC_DISCRETE_BEAM 19.172
*MAT_SID_DAMPER_DISCRETE_BEAM 19.177
*MAT_HYDRAULIC_GAS_DAMPER_DISCRETE_BEAM 19.182
*MAT_CABLE_DISCRETE_BEAM 19.185
*MAT_BILKHU/DUBOIS_FOAM 19.187
*MAT_GENERAL_VISCOELASTIC 19.189
*MAT_HYPERELASTIC_RUBBER 19.193
*MAT_OGDEN_RUBBER 19.197

TABLE OF CONTENTS

*MAT_SOIL_CONCRETE 19.200
*MAT_HYSTERETIC_SOIL 19.204
*MAT_PLASTICITY_WITH_DAMAGE 19.207
*MAT_ORTHOTROPIC_VISCOELASTIC 19.210
*MAT_CELLULAR_RUBBER 19.213
*MAT_ACOUSTIC 19.218
*MAT_SPRING_ELASTIC 19.221
*MAT_DAMPER_VISCOUS 19.222
*MAT_SPRING_ELASTOPLASTIC 19.223
*MAT_SPRING_NONLINEAR_ELASTIC 19.224
*MAT_DAMPER_NONLINEAR_VISCOUS 19.225
*MAT_SPRING_GENERAL_NONLINEAR 19.226
*MAT_SPRING_MAXWELL 19.229
*MAT_SPRING_INELASTIC 19.231
*MAT_SEATBELT 19.232
*MAT_THERMAL_OPTION 19.234
*MAT_THERMAL_ISOTROPIC 19.235
*MAT_THERMAL_ORTHOTROPIC 19.236
*MAT_THERMAL_ISOTROPIC_TD 19.238
*MAT_THERMAL_ORTHOTROPIC_TD 19.240
*MAT_THERMAL_ISOTROPIC_PHASE_CHANGE 19.243
*MAT_THERMAL_ISOTROPIC_TD_LC 19.246

***NODE 20.1**
*NODE 20.1

***PART 21.1**
*PART_OPTION 21.1

***RIGIDWALL 22.1**
*RIGIDWALL_GEOMETRIC_OPTION_{OPTION} 22.2
*RIGIDWALL_PLANAR_{OPTION}_{OPTION}_{OPTION} 22.8

***SECTION 23.1**
*SECTION_BEAM 23.2
*SECTION_DISCRETE 23.6
*SECTION_SEATBELT 23.8
*SECTION_SHELL 23.9
*SECTION_SOLID_OPTION 23.12

*SECTION_TSHELL 23.14

***SET 24.1**

*SET_BEAM 24.2

*SET_DISCRETE 24.3

*SET_NODE_OPTION 24.4

*SET_PART_OPTION 24.6

*SET_SEGMENT 24.9

*SET_SHELL_OPTION 24.11

*SET_SOLID 24.14

*SET_TSHELL 24.15

***TERMINATION 25.1**

*TERMINATION_OPTION 25.1

***TITLE 26.1**

*TITLE 26.1

***TRANSLATE 27.1**

*TRANSLATE_ANSYS_OPTION 27.1

*TRANSLATE_IDEAS_OPTION 27.3

*TRANSLATE_NASTRAN 27.5

***USER 28.1**

*USER_INTERFACE_OPTION 28.1

*USER_LOADING 28.3

RESTART INPUT DATA 29.1

*CHANGE_OPTION 29.3

*CONTROL_DYNAMIC_RELAXATION 29.17

*CONTROL_TERMINATION 29.19

*CONTROL_TIMESTEP 29.20

*DAMPING_GLOBAL 29.21

*DATABASE_OPTION 29.22

*DATABASE_BINARY_OPTION 29.24

*DELETE_OPTION 29.25

*INTERFACE_SPRINGBACK 29.27

*RIGID_DEFORMABLE_OPTION 29.29

*STRESS_INITIALIZATION_OPTION 29.32

*STRESS_INITIALIZATION 29.33

TABLE OF CONTENTS

*STRESS_INITIALIZATION_DISCRETE 29.34
*STRESS_INITIALIZATION_SEATBELT 29.34
*TERMINATION_OPTION 29.35
*TITLE 29.37

REFERENCES 30.1

APPENDIX A

USER DEFINED MATERIALS A.1

APPENDIX B

USER DEFINED AIRBAG SENSOR B.1

APPENDIX C

USER DEFINED SOLUTION CONTROL C.1

APPENDIX D

USER DEFINED INTERFACE CONTROL D.1

APPENDIX E

USER DEFINED INTERFACE FRICTION E.1

APPENDIX F

OCCUPANT SIMULATION INCLUDING THE COUPLING TO PROGRAMS CAL3D AND MADYMO F.1
INTRODUCTION F.1
THE LS-DYNA3D/OCCUPANT SIMULATION PROGRAM LINK F.1
DUMMY MODELING F.4
AIRBAG MODELING F.4
KNEE BOLSTER F.6
COMMON ERRORS F.6

APPENDIX G

INTERACTIVE GRAPHICS COMMANDS G.1

APPENDIX H

INTERACTIVE MATERIAL MODEL DRIVER H.1
INTRODUCTION H.1
INPUT DEFINITION H.1
INTERACTIVE DRIVER COMMANDS H.3

APPENDIX I

VDA DATABASE I.1

APPENDIX J

LS-TAURUS USER'S MANUAL J.1

(To open the LS-TAURUS User's Manual, select LS-TAURUS in the Bookmark List which should be located at the left-hand side of this window.)

LS-DYNA3D USER'S MANUAL (Nonlinear Dynamic Analysis of Structures in Three Dimensions)

ABSTRACT

This manual provides a description of the input data required by Version 93X of LS-DYNA3D. A new keyword database provides a more flexible and logically organized data input scheme. We believe this reorganization will ultimately reduce the time required to understand the input since it eliminates much of confusion of past versions by combining similar functions together under the same keyword. For example, under the keyword *ELEMENT we not only include solid, beam, and shell elements, but also spring elements, discrete dampers, seat belts, and lumped masses. In Version 92X, these elements were specified in separate and disjoint sections of the user's manual. Materials and contact algorithms are specified by names and not by type numbers making the data more readable by those less familiar with the program. Material properties for all elements are defined in one section under the keyword *MAT thereby eliminating three separate sections of material input required by Version 92X. No ordering of the input is expected or required. Either formatted or unformated input may be used with commas serving as delimiters in the latter case. Although the implementation of keyword input meant the complete restructuring of the input phase, we have kept the option of reading the input data prepared for earlier versions of LS-DYNA3D to make the transition in the translators from the structured input file to the keyword file as simple and painless as possible. New capabilities in Version 93X are supported in the structured file so that existing translators to Version 92X can be quickly updated.

This latest revision of LS-DYNA3D (936) has a much improved user's manual plus many new capabilities including:

- Belyschko-Leviathan quadrilateral shell element,
- Automatic rigid to deformable switching,
- Damage based plasticity,
- Trim curves for metal forming springback,
- Multi-chambered airbags and bag-to-bag venting,
- Local coordinate systems for cross-section output,
- Stress initialization for beams, shell, and solid elements,
- More user control for hourglass control constants,
- Table definitions for strain rate effects,

- Coupling with Madymo version 5.1,
- general linear viscoelasticity,
- Ogden rubber model,
- Least squares fit for viscoelastic material constants,
- Implicit heat transfer.

Also, the error checking in LS-DYNA3D has been substantially improved to find input errors before execution begins.

INTRODUCTION

CHRONOLOGICAL HISTORY

DYNA3D [Hallquist 1976] was originated in 1976 at the Lawrence Livermore National Laboratory. The early applications were primarily related to the low velocity impact of heavy, solid structures. These applications tended to be time consuming, and potential users were discouraged by the potentially long run times. Part of the problem of course was related to the rather inefficient implementation of the element technology which was further aggravated by the fact that the super-computer speed in 1976 was less than today's desktop workstation. Furthermore, the primitive sliding interface had only the capability to treat logically regular interfaces that are rather uncommon in most finite element discretizations of complicated three dimensional geometries. This early version of DYNA3D contained truss, membrane, and solid elements. The solid elements ranged from a one point quadrature eight noded element to a twenty noded element with eight point integration. Due to the high cost of the twenty node solid, the zero energy modes related to under integration, and the high frequency content which drove the time step size down, higher order elements were all but abandoned in later versions of DYNA3D.

In an attempt to alleviate these drawbacks, a new version of DYNA3D was released in 1979 that was programmed to provide near optimal speed on the CRAY-1 computers, contained an improved sliding interface treatment that permitted triangular segments, and was an order of magnitude faster than the previous treatment. The 1979 version eliminated structural and higher order solid elements and some of the material models of the first version This version also included an optional element-wise implementation of the integral difference method of Wilkins et al. [1974]. DYNA3D has been used continuously since 1979.

The 1981 version [Hallquist 1981a] evolved from the 1979 version. Nine additional material models were added to allow a much broader range of problems to be modeled including explosive-structure and soil-structure interactions. Body force loads were implemented for angular velocities and base accelerations. A link was also established from the 3D Eulerian code JOY [Couch, et. al., 1983] for studying the structural response to impacts by penetrating projectiles. An option was provided for storing element data on disk thereby doubling the capacity of DYNA3D.

The 1982 version of DYNA3D [Hallquist 1982] accepted DYNA2D [Hallquist 1980] material input directly. The new organization was such that equations of state and constitutive models of any complexity could be easily added. Complete vectorization of the material models had been nearly achieved with about a 10 percent increase in execution speed over the 1981 version.

INTRODUCTION

In the 1986 version of DYNA3D [Hallquist and Benson 1986], many new features were added, including beams, shells, rigid bodies, single surface contact, interface friction, discrete springs and dampers, optional hourglass treatments, optional exact volume integration, and VAX/ VMS, IBM, UNIX, COS operating systems compatibility, that greatly expanded its range of applications. DYNA3D thus became the first code to have a general single surface contact algorithm.

In the 1987 version of DYNA3D [Hallquist and Benson 1987] metalforming simulations and composite analysis became a reality. This version included shell thickness changes, the Belytschko-Tsay shell element [Belytschko and Tsay, 1981], and dynamic relaxation. Also included were non-reflecting boundaries, user specified integration rules for shell and beam elements, a layered composite damage model, and single point constraints.

New capabilities added in the 1988 DYNA3D [Hallquist 1988] version included a cost effective resultant beam element, a truss element, a C0 triangular shell, the BCIZ triangular shell [Bazeley et al. 1965], mixing of element formulations in calculations, composite failure modeling for solids, noniterative plane stress plasticity, contact surfaces with spot welds, tiebreak sliding surfaces, beam surface contact, finite stonewalls, stonewall reaction forces, energy calculations for all elements, a crushable foam constitutive model, comment cards in the input, and one-dimensional slidelines.

In 1988 the author began working half-time at LLNL to devote more time to the development and support of LS-DYNA3D for automotive applications. By the end of 1988 it was obvious that a much more concentrated effort would be required in the development of LS-DYNA3D if problems in crashworthiness were to be properly solved; therefore, at the start of 1989 the author resigned from LLNL to continue code development full time at Livermore Software Technology Corporation. The 1989 version introduced many enhanced capabilities including a one-way treatment of slide surfaces with voids and friction; cross-sectional forces for structural elements; an optional user specified minimum time step size for shell elements using elastic and elastoplastic material models; nodal accelerations in the time history database; a compressible Mooney-Rivlin material model; a closed-form update shell plasticity model; a general rubber material model; unique penalty specifications for each slide surface; external work tracking; optional time step criterion for 4-node shell elements; and internal element sorting to allow full vectorization of right-hand-side force assembly.

During the past four years, considerable progress has been made as may be seen in the chronology of the developments which follows. During 1989 many extensions and developments were completed, and in 1990 the following capabilities were delivered to users:

- arbitrary node and element numbers,
- fabric model for seat belts and airbags,
- composite glass model,
- vectorized type 3 contact and single surface contact,

- many more I/O options,
- all shell materials available for 8 node brick shell,
- strain rate dependent plasticity for beams,
- fully vectorized iterative plasticity,
- interactive graphics on some computers,
- nodal damping,
- shell thickness taken into account in shell type 3 contact,
- shell thinning accounted for in type 3 and type 4 contact,
- soft stonewalls,
- print suppression option for node and element data,
- massless truss elements, rivets – based on equations of rigid body dynamics,
- massless beam elements, spot welds – based on equations of rigid body dynamics,
- expanded databases with more history variables and integration points,
- force limited resultant beam,
- rotational spring and dampers, local coordinate systems for discrete elements,
- resultant plasticity for C^0 triangular element,
- energy dissipation calculations for stonewalls,
- hourglass energy calculations for solid and shell elements,
- viscous and Coulomb friction with arbitrary variation over surface,
- distributed loads on beam elements,
- Cowper and Symonds strain rate model,
- segmented stonewalls,
- stonewall Coulomb friction,
- stonewall energy dissipation,
- airbags (1990),
- nodal rigid bodies,
- automatic sorting of triangular shells into C^0 groups,
- mass scaling for quasi static analyses,
- user defined subroutines,
- warpage checks on shell elements,
- thickness consideration in all contact types,
- automatic orientation of contact segments,
- sliding interface energy dissipation calculations,
- nodal force and energy database for applied boundary conditions,
- defined stonewall velocity with input energy calculations,

and in 1991-1992:

- rigid/deformable material switching,
- rigid bodies impacting rigid walls,
- strain-rate effects in metallic honeycomb model 26,
- shells and beams interfaces included for subsequent component analyses,
- external work computed for prescribed displacement/velocity/accelerations,
- linear constraint equations,
- MPGS database,
- MOVIE database,
- Slideline interface file,
- automated contact input for all input types,
- automatic single surface contact without element orientation,
- constraint technique for contact,
- cut planes for resultant forces,
- crushable cellular foams,
- urethane foam model with hystersis,
- subcycling,
- friction in the contact entities,
- strains computed and written for the 8 node thick shells,
- “good” 4 node tetrahedron solid element with nodal rotations,
- 8 node solid element with nodal rotations,
- 2×2 integration for the membrane element,
- Belytschko-Schwer integrated beam,
- thin-walled Belytschko-Schwer integrated beam,
- improved TAURUS database control,
- null material for beams to display springs and seatbelts in TAURUS,
- parallel implementation on Crays and SGI computers,
- coupling to rigid body codes,
- seat belt capability.

and 1993-1994:

- Arbitrary Lagrangian Eulerian brick elements,
- Belytschko-Wong-Chiang quadrilateral shell element,
- Warping stiffness in the Belytschko-Tsay shell element,
- Fast Hughes-Liu shell element,
- Fully integrated brick shell element,

- Discrete 3D beam element,
- Generalized dampers,
- Cable modeling,
- Airbag reference geometry,
- Multiple jet model,
- Generalized joint stiffnesses,
- Enhanced rigid body to rigid body contact,
- Orthotropic rigid walls,
- Time zero mass scaling,
- Coupling with USA (Underwater Shock Analysis),
- Layered spot welds with failure based on resultants or plastic strain,
- Fillet welds with failure,
- Butt welds with failure,
- Automatic eroding contact,
- Edge-to-edge contact,
- Automatic mesh generation with contact entities,
- Drawbead modeling,
- Shells constrained inside brick elements,
- NIKE3D coupling for springback,
- Barlat's anisotropic plasticity,
- Superplastic forming option,
- Rigid body stoppers,
- Keyword input,
- Adaptivity,
- First MPP (Massively Parallel) version with limited capabilities.
- Built in least squares fit for rubber model constitutive constants,
- Large hystersis in hyperelastic foam,
- Bilhku/Dubois foam model,
- Generalized rubber model,

and many more enhancements not mentioned above.

In the sections that follow, some aspects of the current version of LS-DYNA3D are briefly discussed.

DESCRIPTION OF KEYWORD INPUT

The new keyword input database in Version 93X provides a more flexible and logically organized database that will hopefully reduce the time required by new users in understanding the input. Similar functions are grouped together under the same keyword. For example, under the keyword *ELEMENT we not only include solid, beam, and shell elements, but also spring elements, discrete dampers, seat belts, and lumped masses. In Version 92X, these elements were specified in separate and disjoint sections of the User's Manual. Materials and contact algorithms are specified by names and not by type numbers making the data more readable by those less familiar with the program.

LS-DYNA3D User's Manual is alphabetically organized in logical sections of input data. Each logical section relates to a particular input. There is a control section for resetting LS-DYNA3D defaults, a material section for defining constitutive constants, an equation of state section, an element section where element part identifiers and nodal connectivities are defined, a section for defining parts, and so on. Nearly all model data can be input in block form. For example, consider the following where two nodal points with their respective coordinates and shell elements with their part identity and nodal connectivities are defined:

```
$
$       DEFINE TWO NODES
$
*NODE
        10101       x       y       z
        10201       x       y       z
$
$       DEFINE TWO SHELL ELEMENTS
$
*ELEMENT_SHELL
        10201       pid     n1      n2      n3      n4
        10301       pid     n1      n2      n3      n4
```

Alternatively, acceptable input could also be of the form:

```
$
$       DEFINE  ONE NODE
$
*NODE
        10101       x       y       z
$
$       DEFINE ONE SHELL ELEMENTS
```

```
$
*ELEMENT_SHELL
      10201       pid    n1     n2     n3     n4
$
$     DEFINE  ONE  MORE NODE
$
*NODE
      10201       x      y      z
$
$     DEFINE ONE MORE SHELL ELEMENTS
$
*ELEMENT_SHELL
      10301       pid    n1     n2     n3     n4
```

A data block begins with a keyword followed by data pertaining to the keyword. The next keyword encountered during the reading of the block data defines the end of the block and the beginning of a new block. A keyword must be left justified with the "*" contained in column one. A dollar sign "$" in column one precedes a comment and causes the input line to be ignored. Data blocks are not a requirement for LS-DYNA3D but they can be used to group nodes and elements for user convenience. Multiple blocks can be defined with each keyword if desired as shown above. It would be possible to put all nodal points definitions under one keyword *NODE, or to define one *NODE keyword prior to each node definition. The entire LS-DYNA3D input is order independent with the exception of the optional keyword, *END, which defines the end of input stream. Without the *END termination is assumed to occur when an end-of-file is encountered during the reading.

Figure I.1 attempts to show the general philosophy of the input organization and how various entities relate to each other. In this figure the data included for the keyword, *ELEMENT, is the element identifier, EID, the part identifier, PID, and the nodal points identifiers, the NID's, defining the element connectivity: N1, N2, N3, and N4. The nodal point identifiers are defined in the *NODE section where each NID should be defined just once. A part defined with the *PART keyword has a unique part identifier, PID, a section identifier, SID, a material or constitutive model identifier, MID, an equation of state identifier, EOSID, and the hourglass control identifier, HGID. The *SECTION keyword defines the section identifier, SID, where a section has an element formation specified, a shear factor, SHRF, a numerical integration rule, NIP, and so on. The constitutive constants are defined in the *MAT section where constitutive data is defined for all element types including solids, beams, shells, thick shells, seat belts, springs, and dampers. Equations of state, which are used only with certain *MAT materials for solid elements, are defined in the *EOS section. Since many elements in LS-DYNA3D use uniformly reduced numerical integration, zero energy deformation modes may develop. These modes are controlled numerically

by either an artificial stiffness or viscosity which resists the formation of these undesirable modes. The hourglass control can optionally be user specified using the input in the *HOURGLASS section.

During the keyword input phase where data is read, only limited checking is performed on the data since the data must first be counted for the array allocations and then reordered. Considerably more checking is done during the second phase where the input data is printed out. Since LS-DYNA3D has retained the option of reading older non-keyword input files, we print out the data into the output file D3HSP (default name) as in previous versions of LS-DYNA3D. An attempt is made to complete the input phase before error terminating if errors are encountered in the input. Unfortunately, this is not always possible and the code may terminate with an error message. The user should always check either output file, D3HSP or MESSAG, for the word “Error”.

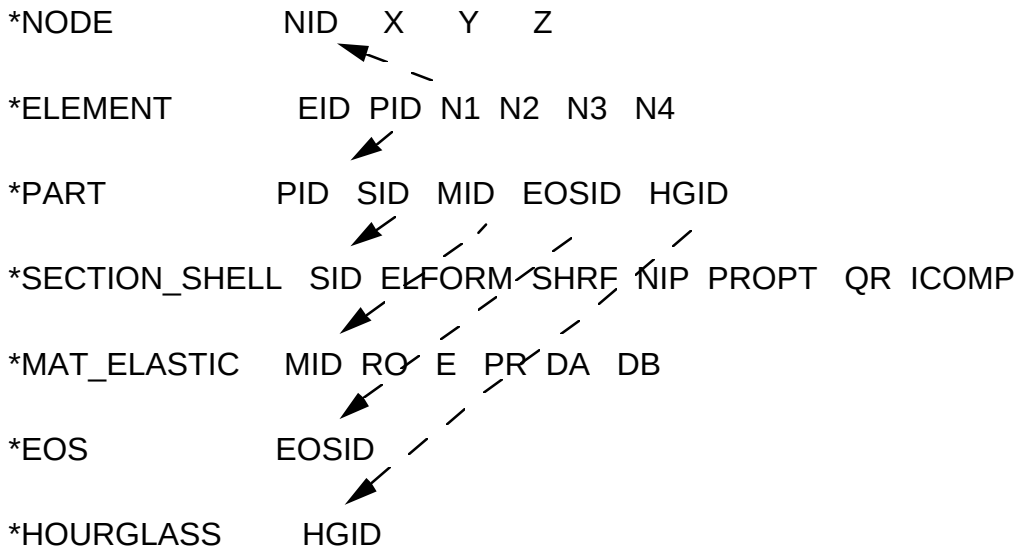

*NODE NID X Y Z
*ELEMENT EID PID N1 N2 N3 N4
*PART PID SID MID EOSID HGID
*SECTION_SHELL SID ELFORM SHRF NIP PROPT QR ICOMP
*MAT_ELASTIC MID RO E PR DA DB
*EOS EOSID
*HOURGLASS HGID

Figure I.1 Organization of the keyword input.

The input data following each keyword can be input in free format. In the case of free format input the data is separated by commas, i.e.,

```
*NODE
10101,x ,y ,z
10201,x ,y ,z
*ELEMENT_SHELL
10201,pid,n1,n2,n3,n4
10301,pid,n1,n2,n3,n4
```

When using commas, the formats **must not** be violated. An I8 integer is limited to a maximum positive value of 99999999, and larger numbers having more than eight characters are unacceptable. The format of the input can change from free to fixed anywhere in the input file. The input is case insensitive and keywords can be given in either upper or lower case. THE ASTERISKS "*" PRECEDING EACH KEYWORD MUST BE IN COLUMN ONE.

To provide a better understanding behind the keyword philosophy and how the options work, a brief review of some of the more important keywords is given below.

***AIRBAG**

The geometric definition of airbags and the thermodynamic properties for the airbag inflator models can be made in this section. This capability is not necessarily limited to the modeling of automotive airbags, but it can also be used for many other applications such as tires and pneumatic dampers.

***BOUNDARY**

This section applies to various methods of specifying either fixed or prescribed boundary conditions. For compatibility with older versions of LS-DYNA3D it is still possible to specify some nodal boundary conditions in the *NODE card section.

***CONSTRAINED**

This section applies constraints within the structure between structural parts. For example, nodal rigid bodies, rivets, spot welds, linear constraints, tying a shell edge to a shell edge with failure, merging rigid bodies, adding extra nodes to rigid bodies and defining rigid body joints are all options in this section.

***CONTACT**

This section is divided in to three main sections. The *CONTACT section allows the user to define many different contact types. These contact options are primarily for treating contact of deformable to deformable bodies, single surface contact in deformable bodies, deformable body to rigid body contact, and tying deformable structures with an option to release the tie base on plastic strain. The surface definition for contact is made up of segments on the shell or solid element surfaces. The keyword options and the corresponding numbers in previous code versions are:

STRUCTURED INPUT TYPE ID	KEYWORD NAME
1	SLIDING_ONLY
p 1	SLIDING_ONLY_PENALTY
2	TIED_SURFACE_TO_SURFACE
3	SURFACE_TO_SURFACE
a 3	AUTOMATIC_SURFACE_TO_SURFACE
4	SINGLE_SURFACE
5	NODES_TO_SURFACE
a 5	AUTOMATIC_NODES_TO_SURFACE
6	TIED_NODES_TO_SURFACE
7	TIED_SHELL_EDGE_TO_SURFACE
8	TIEBREAK_NODES_TO_SURFACE
9	TIEBREAK_SURFACE_TO_SURFACE
10	ONE_WAY_SURFACE_TO_SURFACE
a 10	AUTOMATIC_ONE_WAY_SURFACE_TO_SURFACE
13	AUTOMATIC_SINGLE_SURFACE
a 13	AIRBAG_SINGLE_SURFACE
14	ERODING_SURFACE_TO_SURFACE
15	ERODING_SINGLE_SURFACE
16	ERODING_NODES_TO_SURFACE
17	CONSTRAINT_SURFACE_TO_SURFACE
18	CONSTRAINT_NODES_TO_SURFACE
19	RIGID_BODY_TWO_WAY_TO_RIGID_BODY
20	RIGID_NODES_TO_RIGID_BODY
21	RIGID_BODY_ONE_WAY_TO_RIGID_BODY
22	SINGLE_EDGE
23	DRAWBEAD

The *CONTACT_ENTITY section treats contact between a rigid surface, usually defined as an analytical surface, and a deformable structure. Applications of this type of contact exist in the metalforming area where the punch and die surface geometries can be input as VDA surfaces which are treated as rigid. Another application is treating contact between rigid body occupant dummy hyper-ellipsoids and deformable structures such as airbags and instrument panels. This option is particularly valuable in coupling with the rigid body occupant modeling codes MADYMO and CAL3D. The *CONTACT_1D is for modeling rebars in concrete structure.

***CONTROL**

Options available in the *CONTROL section allow the resetting of default global parameters such as the hourglass type, the contact penalty scale factor, shell element formulation, numerical damping, and termination time.

***DAMPING**

Defines damping either globally or by part identifier.

***DATABASE**

This keyword with a combination of options can be used for controlling the output of ASCII databases and binary files output by LS-DYNA3D. With this keyword the frequency of writing the various databases can be determined.

***DEFINE**

This section allows the user to define curves for loadings, constitutive behaviors, etc.; boxes to limit the geometric extent of certain inputs; local coordinate systems; vectors; and orientation vectors specific to spring and damper elements. Items defined in this section are referenced by their identifiers throughout the input. For example, a coordinate system identifier is sometimes used on the *BOUNDARY cards, and load curves are used on the *AIRBAG cards.

***DEFORMABLE_TO_RIGID**

This section allows the user to switch parts that are defined as deformable to rigid at the start of the analysis. This capability provides a cost efficient method for simulating events such as rollover events. While the vehicle is rotating the computation cost can be reduced significantly by switching deformable parts that are not expected to deform to rigid parts. Just before the vehicle comes in contact with ground, the analysis can be stopped and restarted with the part switched back to deformable.

***ELEMENT**

Define identifiers and connectivities for all elements which include shells, beams, solids, thick shells, springs, dampers, seat belts, and concentrated masses in LS-DYNA3D.

***EOS**

This section reads the equations of state parameters. The equation of state identifier, EOSID, points to the equation of state identifier on the *PART card.

***HOURGLASS**

Defines hourglass and bulk viscosity properties. The identifier, HGID, on the *HOURGLASS card refers to HGID on *PART card.

***INCLUDE**

To make the input file easy to maintain, this keyword allows the input file to be split into subfiles. Each subfile can again be split into sub-subfiles and so on. This option is beneficial when the input data deck is very large.

***INITIAL**

Initial velocity and initial momentum for the structure can be specified in this section. The initial velocity specification can be made by *INITIAL_VELOCITY_NODE card or *INITIAL_ VELOCITY cards. In the case of *INITIAL_VELOCITY_NODE nodal identifiers are used to specify the velocity components for the node. Since all the nodes in the system are initialized to zero, only the nodes with non zero velocities need to be specified. The *INITIAL_VELOCITY card provides the capability of being able to specify velocities using the set concept or boxes.

***INTEGRATION**

In this section the user defined integration rules for beam and shell elements are specified. IRID refers to integration rule number IRID on *SECTION_BEAM and *SECTION_SHELL cards respectively. Quadrature rules in the *SECTION_SHELL and *SECTION_BEAM cards need to be specified as a negative number. The absolute value of the negative number refers to user defined integration rule number. Positive rule numbers refer to the built in quadrature rules within LS-DYNA3D.

***INTERFACE**

Interface definitions are used to define surfaces, nodal lines, and nodal points for which the displacement and velocity time histories are saved at some user specified frequency. This data may then used in subsequent analyses as an interface ID in the *INTERFACE_LINKING_DISCRETE_NODE as master nodes, in *INTERFACE_LINKING_SEGMENT as master segments and in *INTERFACE_LINKING_EDGE as the master edge for a series of nodes. This capability is especially useful for studying the detailed response of a small member in a large structure. For the first analysis, the member of interest need only be discretized sufficiently that the displacements and velocities on its boundaries are reasonably accurate. After the first analysis is completed, the member can be finely discretized in the region bounded by the interfaces. Finally, the second analysis is performed to obtain highly detailed information in the local region of interest. When beginning the first analysis, specify a name for the interface segment file using the Z=parameter on the LS-DYNA3D execution line. When starting the second analysis, the name of the interface segment file created in the first run should be specified using the L=parameter on the LS-DYNA3D command line. Following the above procedure, multiple levels of sub-modeling are easily accommodated. The interface file may contain a multitude of interface definitions so that a single run of a full model can provide enough interface data for many component analyses. The interface feature represents a powerful extension of LS-DYNA3D's analysis capabilities.

***KEYWORD**

Flags LS-DYNA3D that the input deck is a keyword deck. To have an effect this must be the very first card in the input deck. Alternatively, by typing "keyword" on the execute line, keyword input formats are assumed and the "*KEYWORD" is not required. If a number is specified on this card after the word KEYWORD it defines the memory size to used in words. The memory size can also be set on the command line.

***LOAD**

This section provides various methods of loading the structure with concentrated point loads, distributed pressures, body force loads, and a variety of thermal loadings.

***MAT**

This section allows the definition of constitutive constants for all material models available in LS-DYNA3D including springs, dampers, and seat belts. The material identifier, MID, points to the MID on the *PART card.

***NODE**

Define nodal point identifiers and their coordinates.

***PART**

This keyword serves two purposes.

1. Relates part ID to *SECTION, *MATERIAL, *EOS and *HOURGLASS sections.
2. Optionally, in the case of a rigid material, rigid body inertia properties and initial conditions can be specified. Deformable material repositioning data can also be specified in this section if the reposition option is invoked on the *PART card, i.e., *PART_REPOSITION.

***RIGIDWALL**

Rigid wall definitions have been divided into two separate sections, _PLANAR and _GEOMETRIC. Planar walls can be either stationary or moving in translational motion with mass and initial velocity. The planar wall can be either finite or infinite. Geometric walls can be planar as well as have the geometric shapes such as rectangular prism, cylindrical prism and sphere. By default, these walls are stationary unless the option MOTION is invoked for either prescribed translational velocity or displacement. Unlike the planar walls, the motion of the geometric wall is governed by a load curve. Multiple geometric walls can be defined to model combinations of geometric shapes available. For example, a wall defined with the _CYLINDER option can be combined with two walls defined with the _SPHERICAL option to model hemispherical surface caps on the two ends of a cylinder. Contact entities are also analytical surfaces but have the significant advantage that the motion can be influenced by the contact to other bodies, or prescribed with six full degrees-of-freedom.

***SET**

A concept of grouping nodes, elements, materials, etc., in sets is employed throughout the LS-DYNA3D input deck. Sets of data entities can be used for output. So-called slave nodes used

in contact definitions, slaves segment sets, master segment sets, pressure segment sets and so on can also be defined. The keyword, *SET, can be defined in two ways:

1. Option _LIST requires a list of entities, eight entities per card, and define as many cards as needed to define all the entities.

2. Option _COLUMN, where applicable, requires an input of one entity per line along with up to four attribute values which are needed to specify, for example, failure criterion input that is needed for *CONTACT_CONSTRAINT_NODES_TO_SURFACE .

***TITLE**

In this section a title for the analysis is defined.

***USER_INTERFACE**

This section provides a method to provide user control of some aspects of the contact algorithms including friction coefficients via user defined subroutines.

RESTART

This section of the input is intended to allow the user to restart the simulation by providing a restart file and optionally a restart input defining changes to the model such as deleting contacts, materials, elements, switching materials from rigid to deformable, deformable to rigid ,etc.

***RIGID_DEFORMABLE**

This section switches rigid parts back to deformable in a restart to continue the event of a vehicle impacting the ground which may have been modeled with a rigid wall.

***STRESS_INITIALIZATION**

This is an option available for restart runs. In some cases there may be a need for the user to add contacts, elements, etc., which are not available options for standard restart runs. A full input containing the additions is needed if this option is invoked upon restart.

SUMMARY OF COMMONLY USED OPTIONS

The following table gives a list of the commonly used keywords related by topic.

Table I.1. Keywords for the most commonly used options.

Topic	Component	Keyword
Geometry	Nodes	*NODE
	Elements	*ELEMENT_BEAM
		*ELEMENT_SHELL
		*ELEMENT_SOLID
		*ELEMENT_TSHELL
	Discrete Elements	*ELEMENT_DISCRETE
		*ELEMENT_MASS
		ELEMENT_SEATBELT_*Option
Materials	Part (which is composed of Material and Section, equation of state and hourglass data)	*PART
	Material	*MAT_*Option*
	Sections	*SECTION_BEAM
		*SECTION_SHELL
		*SECTION_SOLID
		*SECTION_TSHELL
	Discrete sections	*SECTION_DISCRETE
		*SECTION_SEATBELT
	Equation of state	*EOS_*Option*
	Hourglass	*CONTROL_HOURGLASS
		*HOURGLASS
Contacts and Rigidwalls	Defaults for contacts	*CONTROL_CONTACT
	Definition of contacts	*CONTACT_*Option*
	Definition of rigidwalls	*RIGIDWALL_*Option*

Table I.1. (continued) Keywords for the most commonly used options.

Topic	Component	Keyword
Boundary Conditions & Loadings	Restraints	*NODE
		BOUNDARY_SPC_*Option
	Gravity (body) load	*LOAD_BODY_*Option*
	Point load	*LOAD_NODE_*Option*
	Pressure load	*LOAD_SEGMENT_*Option*
		LOAD_SHELL_*Option
	Thermal load	*LOAD_THERMAL_*Option*
	Load curves	*DEFINE_CURVE
Constraints and spot welds	Constrained nodes	*CONSTRAINED_NODE_SET
	Welds	*CONSTRAINED_GENERALIZED_WELD_ *Option*
		*CONSTRAINED_SPOT_WELD
	Rivet	*CONSTRAINED_RIVET
Output Control	Defaults	*CONTROL_OUTPUT
	ASCII time history files	*DATABASE_*Option*
	Binary plot, time history and restart files	*DATABASE_BINARY_*Option*
	Items in time history blocks	*DATABASE_HISTORY_*Option*
	Nodes for nodal reaction output	*DATABASE_NODAL_FORCE_GROUP
Termination	Termination time	*CONTROL_TERMINATION
	Termination cycle	*CONTROL_TERMINATION
	CPU termination	*CONTROL_CPU
	Degree of freedom	*TERMINATION_NODE

MATERIAL MODELS

Some of the material models presently implemented are:

- elastic,
- orthotropic elastic,
- kinematic/isotropic plasticity [Krieg and Key 1976],
- thermoelastoplastic [Hallquist 1979],
- soil and crushable/non-crushable foam [Key 1974],
- linear viscoelastic [Key 1974],
- Blatz-Ko rubber [Key 1974],
- high explosive burn,
- hydrodynamic without deviatoric stresses,
- elastoplastic hydrodynamic,
- temperature dependent elastoplastic [Steinberg and Guinan 1978],
- isotropic elastoplastic,
- isotropic elastoplastic with failure,
- soil and crushable foam with failure,
- Johnson/Cook plasticity model [Johnson and Cook 1983],
- pseudo TENSOR geological model [Sackett 1987],
- elastoplastic with fracture,
- power law isotropic plasticity,
- strain rate dependent plasticity,
- rigid,
- thermal orthotropic,
- composite damage model [Chang and Chang 1987a 1987b],
- thermal orthotropic with 12 curves,
- piecewise linear isotropic plasticity,
- inviscid, two invariant geologic cap [Sandler and Rubin 1979, Simo et al, 1988a 1988b],
- orthotropic crushable model,
- Mooney-Rivlin rubber,
- resultant plasticity,
- force limited resultant formulation,
- closed form update shell plasticity,
- Frazer-Nash rubber model,

- laminated glass model,
- fabric,
- unified creep plasticity,
- temperature and rate dependent plasticity,
- elastic with viscosity,
- anisotropic plasticity,
- user defined,
- crushable cellular foams (Neilsen, Morgan, and Krieg 1987),
- urethane foam model with hystersis (1992),

and some more foam and rubber models, as well as many materials models for springs and dampers. The hydrodynamic material models determine only the deviatoric stresses. Pressure is determined by one of ten equations of state including:

- linear polynomial [Woodruff 1973],
- JWL high explosive [Dobratz 1981],
- Sack "Tuesday" high explosive [Woodruff 1973],
- Gruneisen [Woodruff 1973],
- ratio of polynomials [Woodruff 1973],
- linear polynomial with energy deposition,
- ignition and growth of reaction in HE [Lee and Tarver 1980, Cochran and Chan 1979],
- tabulated compaction,
- tabulated,
- TENSOR pore collapse [Burton et. al. 1982].

The ignition and growth EOS was adapted from KOVEC [Woodruff 1973]; the other subroutines, programmed by the authors, are based in part on the cited references and are nearly 100 percent vectorized. The forms of the first five equations of state are also given in the KOVEC user's manual and are retained in this manual. The high explosive programmed burn model is described by Giroux [Simo et al. 1988].

The orthotropic elastic and the rubber material subroutines use Green-St. Venant strains to compute second Piola-Kirchhoff stresses, which transform to Cauchy stresses. The Jaumann stress rate formulation is used with all other materials with the exception of one plasticity model which uses the Green-Naghdi rate.

SPATIAL DISCRETIZATION

The elements shown in Figure I.2 are presently available. Currently springs, dampers, beams, membranes, shells, bricks, brick shells and seatbelt elements are included.

The first shell element in DYNA3D was that of Hughes and Liu [Hughes and Liu 1981a, 1981b, 1981c], implemented as described in [Hallquist et al. 1985, Hallquist and Benson 1986]. This element [designated as HL] was selected from among a substantial body of shell element literature because the element formulation has several desirable qualities:

- It is incrementally objective (rigid body rotations do not generate strains), allowing for the treatment of finite strains that occur in many practical applications;
- It is compatible with brick elements, because the element is based on a degenerated brick element formulation. This compatibility allows many of the efficient and effective techniques developed for the DYNA3D brick elements to be used with this shell element;
- It includes finite transverse shear strains;
- A through-the-thickness thinning option (see [Hughes and Carnoy 1981]) is also available.

All shells in our current LS-DYNA3D code must satisfy these desirable traits to at least some extent to be useful in metalforming and crash simulations.

The major disadvantage of the HL element turned out to be cost related and, for this reason, within a year of its implementation we looked at the Belytschko-Tsay [BT] shell [Belytschko and Tsay 1981 1983 1984] as a more cost effective, but possibly less accurate alternative. In the BT shell the geometry of the shell is assumed to be perfectly flat, the local coordinate system originates at the first node of the connectivity, and the co-rotational stress update does not use the costly Jaumann stress rotation. With these and other simplifications, a very cost effective shell was derived which today has become perhaps the most widely used shell elements in both metalforming and crash applications. Results generated by the BT shell usually compare favorably with those of the more costly HL shell. Triangular shell elements are implemented, based on work by Belytschko and co-workers [Belytschko and Marchertas 1974, Bazeley et al. 1965, Belytschko et al. 1984], and are frequently used since collapsed quadrilateral shell elements tend to lock and give very bad results. LS-DYNA3D automatically treats collapsed quadrilateral shell elements as C^0 triangular elements

Since the Belytschko-Tsay element is based on a perfectly flat geometry, warpage is not considered. Although this generally poses no major difficulties and provides for an efficient element, incorrect results in the twisted beam problem and similar situations are obtained where the nodal points of the elements used in the discretization are not coplanar. The Hughes-Liu shell

element considers non-planar geometries and gives good results on the twisted beam. The effect of neglecting warpage in a typical application cannot be predicted beforehand and may lead to less than accurate results, but the latter is only speculation and is difficult to verify in practice. Obviously, it would be better to use shells that consider warpage if the added costs are reasonable and if this unknown effect is eliminated. A new shell has been recently published by Belytschko, Wong, and Chiang [Belytschko, Wong, and Chang 1989, 1992] in which inexpensive modifications were proposed to include the warping stiffness in the Belytschko-Tsay shell. An improved transverse shear treatment also allows the element to pass the Kirchhoff patch test. This element is now available in LS-DYNA3D. Also, two shell elements which use full integration are available in LS-DYNA3D, but are rather expensive.

Three-dimensional plane stress constitutive subroutines are implemented for the shell elements which iteratively updates the stress tensor such that the stress component normal to the shell midsurface is zero. An iterative update is necessary to accurately determine the normal strain component which is necessary to predict thinning. One constitutive evaluation is made for each integration point through the shell thickness.

Zero energy modes in the shell and solid elements are controlled by either an hourglass viscosity or stiffness. Eight node solid-shell elements are implemented and have been found to perform well in many applications. All elements are nearly 100% vectorized. All element classes can be included as parts of a rigid body. The rigid body formulation is documented in [Benson and Hallquist 1986]. Rigid body point nodes, as well as concentrated masses, springs and dashpots can be added to this rigid body.

Membrane elements can be either defined directly as shell elements with a membrane formulation option or as shell elements with only one point for through thickness integration. The latter choice includes transverse shear stiffness and may be inappropriate. For airbag material a special fully integrated three and four node membrane element is available.

Two different beam types are available: a stress resultant beam and a beam with cross section integration at one point along the axis. The cross section integration allows for a more general definition of arbitrarily shaped cross sections taking into account material nonlinearities.

Spring and damper elements can be translational or rotational. Many behavior options can be defined, e.g., arbitrary nonlinear behavior including locking and separation.

Solid elements in LS-DYNA3D may be defined using from 4 to 8 nodes. The standard elements are based on linear shape functions and use one point integration and hourglass control. A selective-reduced integrated (called fully integrated) 8 node solid element is available for situations when the hourglass control fails. Also, two additional solid elements, a 4 noded tetrahedron and an 8 noded hexahedron, with nodal rotational degrees of freedom, are implemented based on the idea of Allman [1984] to replace the nodal midside translational degrees of freedom of the elements with

quadratic shape functions by corresponding nodal rotations at the corner nodes. The latter elements, which do not need hourglass control, require many numerical operations compared to the hourglass controlled elements and should be used at places where the hourglass elements fail. However, it is well known that the elements using more than one point integration are more sensitive to large distortions than one point integrated elements.

The brick shell or solid shell element is a shell element with only nodal translations for the eight nodes. The assumptions of shell theory are included in a non-standard fashion. It also uses hourglass control or selective-reduced integration. This element can be used in place of any four node shell element. It is favorably used for shell-brick transitions, as no additional constraint conditions are necessary. However, care has to be taken to know in which direction the shell assumptions are made; therefore, the numbering of the element is important.

Seatbelt elements can be separately defined to model seatbelt actions combined with dummy models. Separate definitions of seatbelts, which are one-dimensional elements, with accelerometers, sensors, pretensioners, retractors, and sliprings are possible. The actions of the various seatbelt definitions can also be arbitrarily combined.

SLIDING INTERFACES

The three-dimensional contact-impact algorithm was originally an extension of the NIKE2D [Hallquist 1979] two-dimensional algorithm. As currently implemented, one surface of the interface is identified as a master surface and the other as a slave. Each surface is defined by a set of three or four node quadrilateral segments, called master and slave segments, on which the nodes of the slave and master surfaces, respectively, must slide. In general, an input for the contact-impact algorithm requires that a list of master and slave segments be defined. For the single surface algorithm only the slave surface is defined and each node in the surface is checked each time step to ensure that it does not penetrate through the surface. Internal logic [Hallquist 1977, Hallquist et al. 1985] identifies a master segment for each slave node and a slave segment for each master node and updates this information every time step as the slave and master nodes slide along their respective surfaces. It must be noted that for general automatic definitions only parts/materials or three-dimensional boxes have to be given. Then the possible contacting outer surfaces are identified by the internal logic in LS-DYNA3D. More than 20 types of interfaces can presently be defined including:

sliding only for fluid/structure or gas/structure interfaces,
tied,
sliding, impact, friction,
single surface contact,
discrete nodes impacting surface,
discrete nodes tied to surface,
shell edge tied to shell surface,
nodes spot welded to surface,
tiebreak interface,
one way treatment of sliding, impact, friction,
box/material limited automatic contact for shells,
automatic contact for shells (no additional input required),
automatic single surface with beams and arbitrary orientations,
surface to surface eroding contact,
node to surface eroding contact,
single surface eroding contact,
surface to surface symmetric constraint method [Taylor and Flanagan 1989],
node to surface constraint method [Taylor and Flanagan 1989],
rigid body to rigid body contact with arbitrary force/deflection curve,
rigid nodes to rigid body contact with arbitrary force/deflection curve,

edge-to-edge,
draw beads.

Interface friction can be used with most interface types. The tied and sliding only interface options are similar to the two-dimensional algorithm used in LS-DYNA2D [Hallquist 1976, 1978, 1980]. Unlike the general option, the tied treatments are not symmetric; therefore, the surface which is more coarsely zoned should be chosen as the master surface. When using the one-way slide surface with rigid materials, the rigid material should be chosen as the master surface.

For geometric contact entities, contact has to be separately defined. It must be noted that for the contact of a rigid body with a flexible body, either the sliding interface definitions as explained above or the geometric contact entity contact can be used. Currently, the geometric contact entity definition is recommended for metalforming problems due to high accuracy and computational efficiency.

INTERFACE DEFINITIONS FOR COMPONENT ANALYSIS

Interface definitions for component analyses are used to define surfaces, nodal lines, or nodal points for which the displacement and velocity time histories are saved at some user specified frequency. This data may then used in subsequent analyses as master surfaces of type TIED_SURFACE_TO_SURFACE sliding interfaces of Section 31, as master lines in the tie breaking shell definitions, or as the controlling nodes for determining the motion of single nodal points. This capability is especially useful for studying the detailed response of a small member in a large structure. For the first analysis, the member of interest need only be discretized sufficiently that the displacements and velocities on its boundaries are reasonably accurate. After the first analysis is completed, the member can be finely discretized and interfaces defined to correspond with the first analysis. Finally, the second analysis is performed to obtain highly detailed information in the local region of interest.

When starting the analysis, specify a name for the interface segment file using the Z = parameter on the LS-DYNA3D command line. When starting the second analysis, the name of the interface segment file (created in the first run) should be specified using the L = parameter on the LS-DYNA3D command line.

Following the above procedure, multiple levels of sub-modeling are easily accommodated. The interface file may contain a multitude of interface definitions so that a single run of a full model can provide enough interface data for many component analyses. The interface feature represents a powerful extension of LS-DYNA3D's analysis capability.

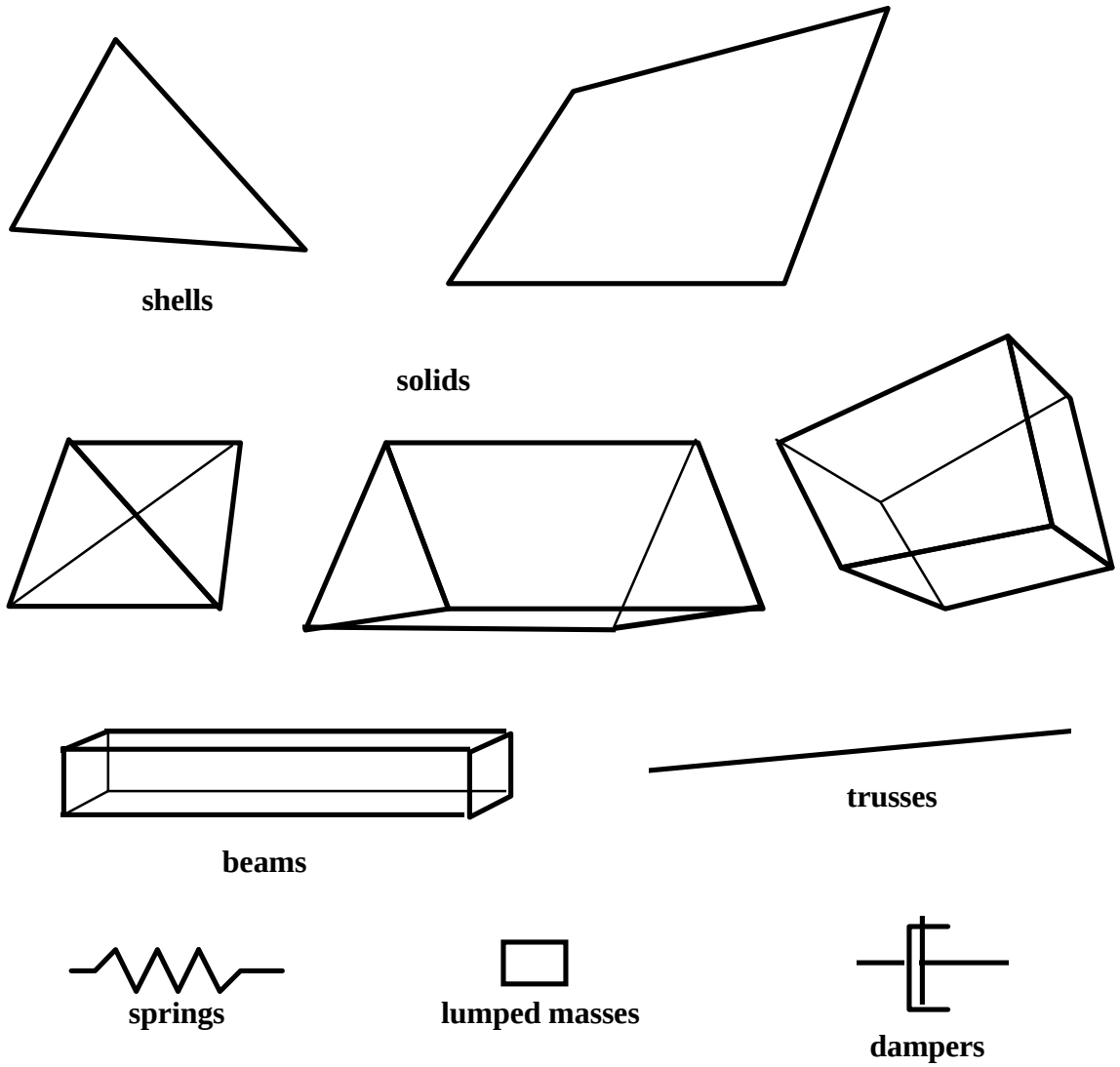


Figure I.2. Elements in LS-DYNA3D.

CAPACITY

Storage allocation is dynamic. The only limit that exists on the number of boundary condition cards, number of material cards, number of pressure cards, etc., is the capacity of the computer. Typical LS-DYNA3D calculations may have 10,000 to 200,000 elements. Memory allocation is dynamic and can be controlled during execution.

CODE ORGANIZATION

LS-DYNA3D consists of one source that compiles under FORTRAN compilers on most UNIX workstations and supercomputers. The programming follows the FORTRAN 77 standard with some parts programmed in C. LS-DYNA3D has eight segments in the main code. They are:

- input,
- restart,
- initialization,
- solution,
- interactive real time graphics,
- rezoning,
- remapping,
- thermal.

Parallel versions of LS-DYNA3D for shared memory are supported for the SGI and CRAY computers, and a distributed memory version of LS-DYNA3D has been ported to a subset of the commercially available MPP machines incuding the CRAY-T3D, IBM SP1/SP2, and the INTEL PARAGON.

SENSE SWITCH CONTROLS

The status of an in-progress LS-DYNA3D simulation can be determined by using the sense switch. On UNIX versions, this is accomplished by first typing a "^C" (Control-C). This sends an interrupt to LS-DYNA3D which is trapped and the user is prompted to input the sense switch code. LS-DYNA3D has five terminal sense switch controls that are tabulated below:

Type	Response
SW1.	A restart file is written and LS-DYNA3D terminates.
SW2.	LS-DYNA3D responds with time and cycle numbers.
SW3.	A restart file is written and LS-DYNA3D continues.
SW4.	A plot state is written and LS-DYNA3D continues.
SW5.	Enter interactive graphics phase (most installations).

On UNIX systems the sense switches can still be used if the job is running in the background or in batch mode. To interrupt LS-DYNA3D simply type "kill -2 psid". LS-DYNA3D will first look for a file called "switch" which should contain the sense switch data. Otherwise, an SW2 is assumed and the output is sent to standard out.

When LS-DYNA3D terminates, all scratch files are destroyed: the restart file, plot files, and high-speed printer files remain on disk. Of these, only the restart file is needed to continue the interrupted analysis.

PRECISION

The explicit time integration algorithms used in LS-DYNA3D are in general much less sensitive to machine precision than other finite element solution methods. Consequently, double precision is not used. The benefits of this are greatly improved utilization of memory and disk. When problems have been found we have usually been able to overcome them by reorganizing the algorithm or by converting to double precision locally in the subroutine where the problem occurs. A few of the known problems include: **(32-bit computers only!):**

- Round-off errors can cause difficulties with extremely small deflection problems. (Maximum vibration amplitudes are $<10^{-6}$ times nodal coordinates).
  Workaround: Increase the load.
- Buckling problems, which are very sensitive to small imperfections.

However, the users of LS-DYNA3D have to be aware of potential problems.

A major reorganization of LS-DYNA3D has led to a version using double precision throughout the full program. As memory and disk space of the computers is less of a problem, we prefer to provide this version for all machines. It also allows LS-DYNA3D to take advantage of the 64-bit technology offered by some computer manufacturers.

EXECUTION SYNTAX

The interactive execution line for LS-DYNA3D is as follows:

LS-DYNA3D I=**inf** O=**otf** G=**ptf** D=**dpf** F=**thf** U=**xtf** T=**tpf** A=**rrd** M=**sif** J=**jif** S=**iff** Z=**isf1** L=**isf2** B=**rlf** W=**root** E=**efl** X=**scl** C=**cpu** K=**kill** V=**vda** Y=**c3d KEYWORD MEMORY=nwds** where

inf	=	input file (user specified),
otf	=	high speed printer file (default=D3HSP),
ptf	=	binary plot file for graphics (default=D3PLOT),
dpf	=	dump file for restarting (default=D3DUMP),
thf	=	binary plot file for time histories of selected data (default=D3THDT),
xtf	=	binary plot file for time extra data (default-XTFILE)
tpf	=	optional temperature file (TOPAZ3D plotfile),
rrd	=	running restart dump file (default=RUNRSF),
sif	=	stress initialization file (user specified),
jif	=	optional JOY interface file,
iff	=	interface force file (user specified),
isf1	=	interface segment save file to be created (user specified),
isf2	=	existing interface segment save file to be used (user specified),
rlf	=	binary plot file for dynamic relaxation (default=D3DRFL),
efl	=	echo file containing optional input echo with or without node/element data,
root	=	root file name for general print option,
scl	=	scale factor for binary file sizes (default=7),
cpu	=	cpu limit in seconds, applies to total calculation not just cpu from a restart,
kill	=	if LS-DYNA3D encounters this file name it will terminate with a restart file (default=D3KIL),
vda	=	VDA/IGES database for geometrical surfaces,
c3d	=	CAL3D input file,
nwds	=	Number of words to be allocated. On engineering workstations a word is usually 32bits.

In order to avoid undesirable results, each LS-DYNA3D run should be performed in a separate directory. Also, files should be removed or renamed to avoid confusion. By including **KEYWORD** anywhere on the execute line or instead if ***KEYWORD** is the first card in the input file, the keyword formats are expected; otherwise, the older structured input file will be expected.

File Organization

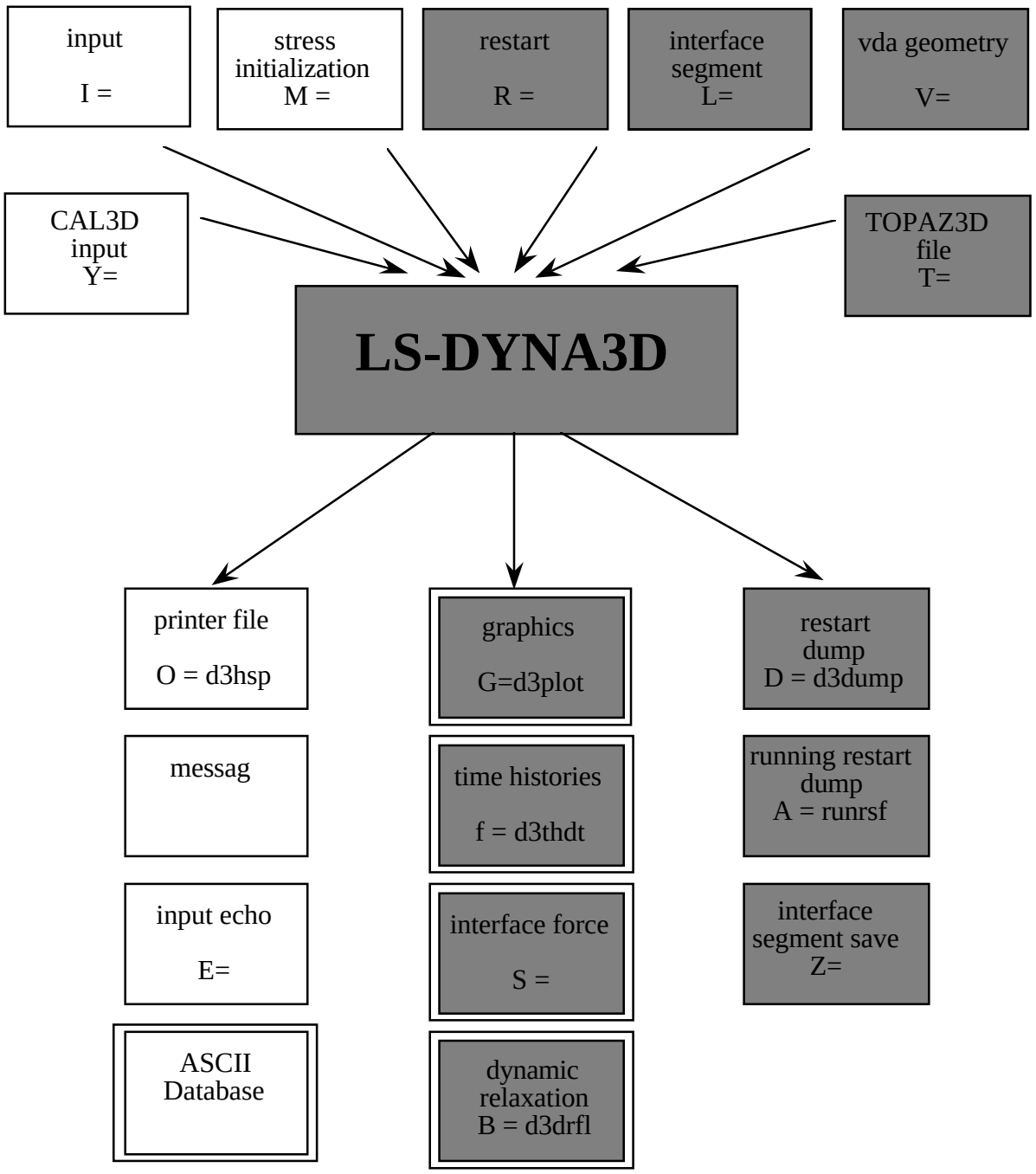


Figure I.3

If the word **MEMORY** is found anywhere on the execution line and if it is not set via (**=nwds**) LS-DYNA3D will give the default size of memory, request, and then read in the desired memory size. This option is necessary if the default value is insufficient memory and termination occurs as a result. Occasionally, the default value is too large for execution and this option can be used to lower the default size.

File names must be unique. The interface force file is created only if it is specified on the execution line (S=iff). On large problems the default file sizes may not be large enough for a single file to hold either a restart dump or a plot state. Then the file size may be increased by specifying the file size on the execute line using X=**scl**. The default file size holds seven times one-million octal word (262144) or 1835008 words. If the core required by LS-DYNA3D requires more space, it is recommended that the **scl** be increased appropriately. Using C=**cpu** defines the maximum cpu usage allowed that if exceeded will cause LS-DYNA3D to terminate with a restart file. During a restart, **cpu** should be set to the total cpu used up to the current restart plus whatever amount of additional time is wanted.

When restarting from a dump file, the execution line becomes

LS-DYNA3D I=**inf** O=**otf** G=**ptf** D=**dpf** R=**rtf** F=**thf** U=**xtf** T=**tpf** A=**rrd** J=**jif** S=**iff** Z=**isf1** L=**isf2** B=**rlf** W=**root** E=**efl** X=**scl** C=**cpu** K=**kill** Q=*option* **KEYWORD MEMORY=nwds**

where

rtf = restart filename.

If the data from the last run is to be remapped onto a new mesh, then specify: Q=**remap.** The remap file is the dump file from which the remapping data are taken. The remap option is available for brick elements only. File name dropouts are permitted; for example, the execution lines are acceptable.

LS-DYNA3D I=**inf**
LS-DYNA3D R=**rtf**

Default names for the output file, binary plot files, and the dump file are D3HSP, D3PLOT, D3THDT, and D3DUMP, respectively.

For an analysis using interface segments the execution line in the first analysis is given by:

LS-DYNA3D **I=inf Z=isf1**

and in the second by:

LS-DYNA3D **I=inf L=isf1**

Batch execution in some installations (e.g., GM) is controlled by file NAMES on unit 88. NAMES is a 2 line file in which the second line is blank. The first line of NAMES contains the execution line:

I=inf

if this is the initial run. For a restart the execution line becomes:

I=inf R=rtf

Remark: No stress initialization is possible at restart. Also the VDA files and the CAL3D files cannot be changed.

RESTART ANALYSIS

The LS-DYNA3D restart capability allows analyses to be broken down into stages. After the completion of each stage in the calculation a "restart dump" is written that contains all information necessary to continue the analysis. The size of this "dump" file is roughly the same size as the memory required for the calculation. Results can be checked at each stage by post-processing the output databases in the normal way, so the chance of wasting computer time on incorrect analyses is reduced. The restart capability is frequently used to modify models by deleting excessively distorted elements, materials that are no longer important, and contact surfaces that are no longer needed. Output frequencies of the various databases can also be altered. Often, these simple modifications permit the calculation to continue on to a successful completion. Restarting can also help to diagnose why a model is giving problems. By restarting from a dump that is written before the occurrence of a numerical problem and obtaining output at more frequent intervals, it is often possible to identify where the first symptoms appear and what aspect of the model is causing them.

The format of the restart input file is described in this manual. If, for example, the user wishes to restart the analysis from dump state *nn*, contained in file *D3DUMPnn*, then the following procedure is followed:

1. Create the restart input deck, if required, as described in the Restart Section of this manual. Call this file *restartinput.*
2. By invoking the execution line:

LS-DYNA3D I=*restartinput* R=*D3DUMPnn*

execution begins. If no alterations to the model are made, then the execution line:

LS-DYNA3D R=*D3DUMPnn*

will suffice. Of course, the other output files should be assigned names if the defaults have been changed in the original run.

The R=D3DUMPnn on the status line informs the program that this is a restart analysis.

The full deck restart option allows the user to begin a new analysis, with deformed shapes and stresses carried forward from a previous analysis for selected materials. The new analysis can be different from the original, e.g., more contact surfaces, different geometry (of parts which are not carried forward), etc. Examples of applications include:

- Crash analysis continued with extra contact surfaces;
- Sheet metalforming continued with different tools for modeling a multi-stage forming process.

Assume an analysis is run using the input file, job1.inf, and a restart dump named d3dump01 is created. A new input file job2.inf is generated and submitted as a restart with R=d3dump01 as the dump file. The input file job2.inf contains the entire model in its original undeformed state but with more contact surfaces, new output databases, and so on. Since this is a restart job, information must be given to tell LS-DYNA3D which parts of the model should be initialized in the full deck restart. When the calculation begins the restart database contained in the file d3dump01 is read, and a new database is created to initialize the model in the input file, job2.inf. The data in file job2.inf is read and the LS-DYNA3D proceeds through the entire input deck and initialization. At the end of the initialization process, all the parts selected are initialized from the data saved from d3dump01. This means that the deformed position and velocities of the nodes on the elements of each part, and the stresses and strains in the elements (and, if the material of the part is rigid, the rigid body properties) will be assigned.

It is assumed during this process that any initialized part has the same elements, in the same order, with the same topology, in job1 and job2. If this is not the case, the parts cannot be initialized. However, the parts may have different identifying numbers.

For discrete elements and seat belts, the choice is all or nothing. All discrete and belt elements, retractors, sliprings, pretensioners and sensors must exist in both files and will be initialized.

Materials which are not initialized will have no initial deformations or stresses. However, if initialized and non-initialized materials have nodes in common, the nodes will be moved by the initialized material causing a sudden strain in the non-initialized material. This effect could give rise to sudden spikes in loading.

Points to note are:

- Time and output intervals are continuous with job1, i.e., the time is not reset to zero.
- Don’t try to use the restart part of the input to change anything since this will be overwritten by the new input file.
- Usually, the complete input file part of job2.in1 will be copied from job1.inf, with the required alterations. We again mention that there is no need to update the nodal coordinates since the deformed shapes of the initialized materials will be carried forward from job1.
- Completely new databases will be generated with the time offset.

VDA/IGES DATABASES

VDA surfaces are surfaces of geometric entities which are given in the form of polynomials. The format of these surfaces is as defined by the German automobile and supplier industry in the VDA guidelines, [VDA, 1987].

The advantage of using VDA surfaces is twofold. First, the problem of meshing the surface of the geometric entities is avoided and, second, smooth surfaces can be achieved which are very important in metalforming. With smooth surfaces, artificial friction introduced by standard faceted meshes with corners and edges can be avoided. This is a big advantage in springback calculations.

A very simple and general handling of VDA surfaces is possible allowing arbitrary motion and generation of surfaces. For a detailed description, see Appendix I.

MESH GENERATION

LS-DYNA3D is designed to operate with a variety of commercial pre-processing packages. Currently, direct support is available from PATRAN[1], FEMB, HYPERMESH, and MEDINA. Several third-party translation programs are available for PATRAN and IDEAS.

Alternately, the pre-processor LS-INGRID [LSTC Report 1019] is available from LSTC and is specialized to LS-DYNA3D. Some of the capabilities available in LS-INGRID are:

Complete support for all control parameters, loads and material types,
Mass property calculations,
Importing models from other sources (PATRAN, IDEAS, IGES and NASTRAN formats),
Interactive viewing and graphical inspection of boundary conditions, etc.,
Model editing,
General purpose mesh generation,
Importing LS-DYNA3D and DYNA3D models in a variety of older formats,
Complex surface treatment including NURB surfaces,
Parametric modeling.

Capabilities specialized to automotive applications:

Airbag folding and inspection,
Occupant positioning,
Seat belt positioning (both beam and shells),
Merging of occupants, airbags and belts with car models.

[1] PATRAN is a trademark of PDA Engineering, HYPERMESH is a trademark of Altair Engineering, FEMB is a trademark of Engineering Technology Associates, IDEAS is a trademark of Structural Dynamics Research Corporation.

TAURUS POST-PROCESSING

LS-TAURUS [Brown and Hallquist 1984] processes output from LS-DYNA3D. LS-TAURUS reads the binary plot-files generated by LS-DYNA3D and plots contours, fringes, time histories, and deformed shapes. Color contours and fringes of a large number of quantities may be interactively plotted on meshes consisting of plate, shell, and solid type elements. LS-TAURUS can compute a variety of strain measures, reaction forces along constrained boundaries, and momenta. LS-TAURUS is operational on the CRAY, VAX, SUN, APOLLO, IBM RS6000, SGI, STARDENT, HP and MIPS computers. Interfaces from LS-TAURUS to other commercial post-processors are available.

LS-DYNA3D generates three binary databases. One contains information for complete states at infrequent intervals; 50 to 100 states of this sort is typical in a LS-DYNA3D calculation. The second contains information for a subset of nodes and elements at frequent intervals; 1000 to 10,000 states is typical. The third contains interface data for contact surfaces.

Because of the difficulty in handling one large file, an alternative method for obtaining printed output is also available. Many ASCII databases are created at the user's option containing such information as cross-sectional forces, rigidwall forces, nodal point data, element integration point data, global data like total internal and kinetic energy, material energies, nodal interface forces, resultant interface forces, single point constraint forces, as well as files that are compatible with MOVIE.BYU and the Cray Research developed post-processor, MPGS. A SMUG animator database and a NASTRAN BDF file is written for users at General Motors. Each ASCII database is written at its own unique output interval defined in the user input.

File Organization

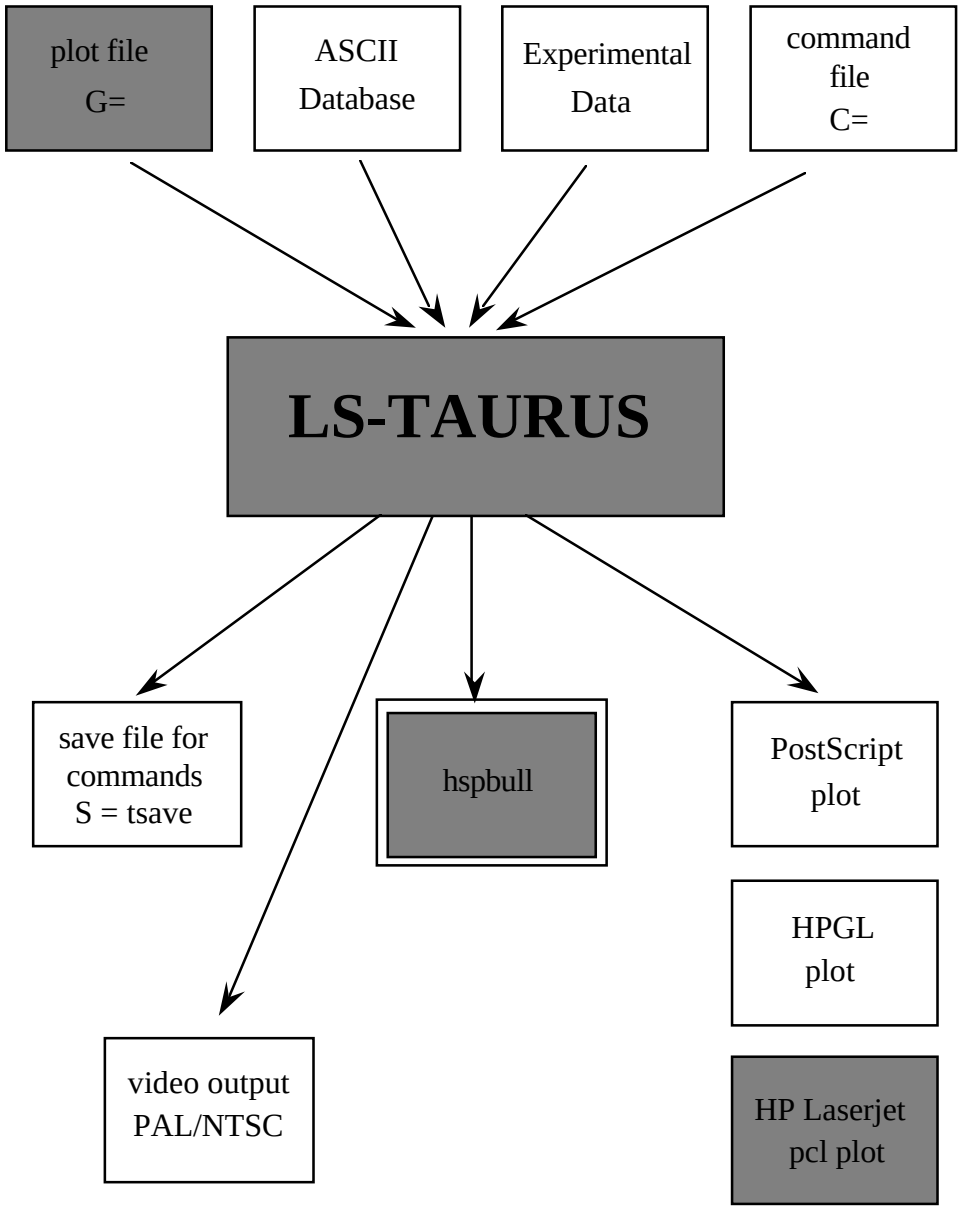


Figure I.4

EXECUTION SPEEDS

The execution speeds on the Cray-YMP for various elements in LS-DYNA3D are tabulated below in microseconds per element cycle:

Element Type	CPU Cost
8 node solid with 1 point integration and default hourglass control	12
as above but with Flanagan-Belytschko hourglass control	15
constant stress and Flanagan-Belytschko hourglass control, i.e., the Flanagan-Belytschko element	20
4 node Belytschko-Tsay shell with four thickness integration points	11
4 node Belytschko-Tsay shell with resultant plasticity	9
BCIZ triangular shell with four thickness integration points	22
C^0 triangular shell with four thickness integration points	11
2 node Hughes-Liu beam with four integration points	28
2 node Belytschko-Schwer beam	5
2 node simple truss elements	3
8 node solid-shell with four through the thickness integration points	33

These timings are very approximate and do not account for the inclusion of sliding interfaces or complex material models. Each interface node of the sliding interfaces is roughly equivalent to one-half zone cycle in cost. Figure I.5 illustrates the relative cost of the various shell formulations in LS-DYNA3D.

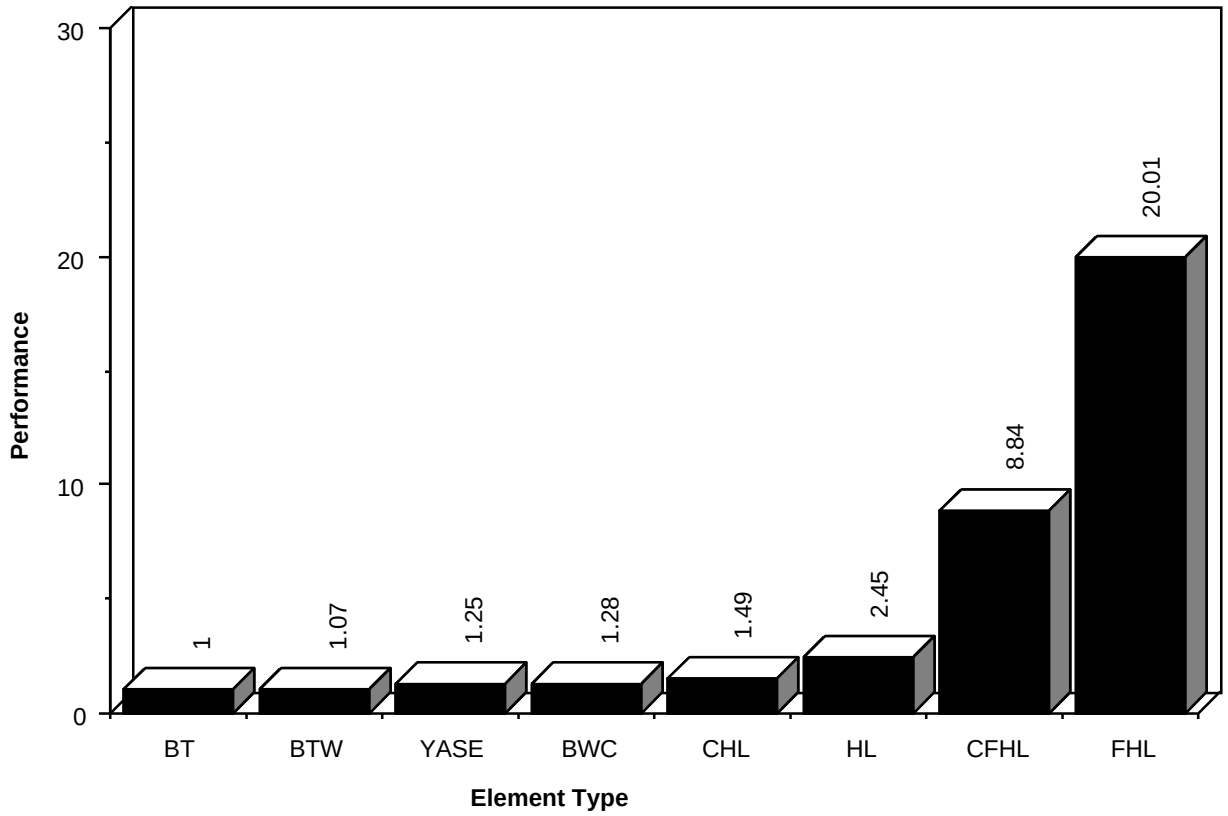


Figure I.5. Relative cost of the four noded shells available in LS-DYNA3D where BT is the Belytschko-Tsay shell, BTW is the Belytschko-Tsay shell with the warping stiffness taken from the Belytschko-Wong-Chiang, BWC, shell. The YASE shell is the Englemann-Whirley shell. CHL denotes the Hughes-Liu shell, HL, with one point quadrature and a co-rotational formulation. FHL is the fully integrated Hughes-Liu shell, and the CFHL shell is its co-rotational version.

UNITS

The units in LS-DYNA3D must be consistent. One way of testing whether a set of units is consistent is to check that:

1 (force unit) = 1 (mass unit) x 1 (acceleration unit)

and that 1 (acceleration unit) = $\frac{1(\text{length unit})}{\left[1(\text{time unit})\right]^2}$

Examples of sets of consistent units are:

	(a)	(b)	(c)
Length unit	meter	millimeter	millimeter
Time unit	second	second	millisecond
Mass unit	kilogram	tonne	kilogram
Force unit	Newton	Newton	kiloNewton
Young's Modulus of Steel	210.0E+09	210.0E+03	210.0
Density of Steel	7.85E+03	7.85E–09	7.85E–06
Yield stress of Mild Steel	200.0E+06	200.0	0.200
Acceleration due to gravity	9.81	9.81E+03	9.81E–03
Velocity equivalent to 30 mph	13.4	13.4E+03	13.4

GENERAL CARD FORMAT

The following sections specify for each keyword the cards that have to be defined. Each card is defined in its rigid format form and is shown as a number of fields in an 80 character string. **Most cards are 8 fields with a length of 10 and a sample card is shown below.**

Card Format

	1	2	3	4	5	6	7	8
Variable	NSID	PSID	A1	A2	A3	SASH		
Type	I	I	F	F	F	I		
Default	none	none	1.0	1.0	0	1		
Remarks	1			2		3		

The type is the variable type and is either F, for floating point or I, for an integer. The default gives the value set if zero is specified, the field is left blank or the card is not defined. The remarks refer to comments at the end of the section. The card format is given above the card if it is other than eight fields of 10. Free formats may be used with the data separated by commas. When using comma format, the number of characters used to specify a number must not exceed the number which would fit into the equivalent rigid format field. An I8 number is limited to a number of 99999999 and larger numbers with more than eight characters are unacceptable. Rigid and free formats can be mixed throughout the deck but not within a card.

MPP/LS-DYNA3D USER INFORMATION

Supported Features

First and foremost, the only input formats currently supported are 920, 930, and keyword. Models in any of the older formats will need to be converted to one these input formats before then can be run with the current version of LS-DYNA3D for massively parallel processors, mpp.

The large majority of LS-DYNA3D options are available on the MPP computers. Those that are not supported are being systematically added. Unless otherwise noted here, all options of LS-DYNA3D version 93*x* are supported by MPP/LS-DYNA3D.

Here is the list of unsupported options:

- *ALE
- *BOUNDARY_CONVECTION
- *BOUNDARY_CYCLIC
- *BOUNDARY_FLUX
- *BOUNDARY_RADIATION
- *BOUNDARY_TEMPERATURE
- *BOUNDARY_USA_SURFACE
- *CONSTRAINED_RIGID_BODY_STOPPERS
- *CONSTRAINED_SHELL_TO_SOLID
- *CONSTRAINED_TIE-BREAK
- *CONSTRAINED_TIED_NODES_FAILURE
- *CONTROL_ADAPTIVE
- *CONTACT_AIRBAG_SINGLE_SURFACE
- *CONTACT_CONSTRAINT_NODES_TO_SURFACE
- *CONTACT_CONSTRAINT_SURFACE_TO_SURFACE
- *CONTACT_DRAWBEAD
- *CONTACT_ERODING_NODES_TO_SURFACE
- *CONTACT_ERODING_SINGLE_SURFACE
- *CONTACT_ERODING_SURFACE_TO_SURFACE
- *CONTACT_RIGID_NODES_TO_RIGID_BODY
- *CONTACT_RIGID_BODY_ONE_WAY_TO_RIGID_BODY
- *CONTACT_RIGID_BODY_TWO_WAY_TO_RIGID_BODY
- *CONTACT_SINGLE_EDGE
- *CONTACT_SLIDING_ONLY

- *CONTACT_SLIDING_ONLY_PENALTY
- *CONTACT_TIEBREAK_NODES_TO_SURFACE
- *CONTACT_TIEBREAK_SURFACE_TO_SURFACE
- *CONTACT_TIED_SHELL_EDGE_TO_SURFACE
- *CONTACT_1D
- *DATABASE_AVS
- *DATABASE_MOVIE
- *DATABASE_MPGS
- *DATABASE_BEAM, BEAM_SET
- *DATABASE_SOLID, SOLID_SET
- *DATABASE_TSHELL, TSHELL_SET
- *DATABASE_NODAL_FORCE_GROUP
- *DATABASE_TRACER
- *DEFORMABLE_TO_RIGID
- *ELEMENT_SEATBELT_ACCELEROMETER
- *INTERFACE_COMPONENT
- *INTERFACE_JOY
- *LOAD_SUPERPLASTIC_OPTION
- *USER
- *PART_REPOSITION
- *RIGIDWALL_PLANAR_FORCES
- *TERMINATION

MPP/LS-DYNA3D can restart; however, the restart options are still quite limited. Only the termination time, plot interval, time step control, and restart dump frequency may be changed when restarting. The supported keywords are:

- *CONTROL_TERMINATION
- *CONTROL_TIMESTEP
- *DATABASE_BINARY

Arbitrary Numbering

MPP/LS-DYNA3D assumes arbitrarily numbered input. This affects the format of some of the input options. For example, if the initial velocity option is used, all nodes in the problem must appear in the initial velocity section.

Contact Interfaces

MPP/LS-DYNA3D uses a completely redesigned, highly parallel contact algorithm. The contact options currently supported include:

- *CONTACT_AUTOMATIC_NODES_TO_SURFACE
- *CONTACT_AUTOMATIC_ONE_WAY_SURFACE_TO_SURFACE
- *CONTACT_AUTOMATIC_SINGLE_SURFACE
- *CONTACT_AUTOMATIC_SURFACE_TO_SURFACE
- *CONTACT_NODES_TO_SURFACE
- *CONTACT_ONE_WAY_SURFACE_TO_SURFACE
- *CONTACT_SINGLE_SURFACE
- *CONTACT_SURFACE_TO_SURFACE
- *CONTACT_TIED_NODES_TO_SURFACE
- *CONTACT_TIED_SURFACE_TO_SURFACE

Due to the nature of the algorithm it is desirable for all contact materials to have a proper positive thickness associated with them. This is not required, however, insofar as the oriented contact segment options have been implemented. That is to say, for contact types other than 4 and 13 (which always require thickness), contact surface orientation will be used if SHLTHK=0 on the *CONTROL_CONTACT keyword card (Also see Control Card 15, columns 26-30, of the structured input manual.). This allows for 0 thickness contact interfaces. If contact thickness is considered then negative thicknesses can be applied via the Sliding Interface control cards (although this not recommend). In order for the automatic orientation to work properly, the master and slave sides of the contact interface should start in a near contact position (such as most material forming problems). If the surfaces are initially far apart, it is the responsibility of the user to properly orient the contact segments. (This restriction may be relaxed in a later version).

Pre-decomposition

There is an optional auxiliary serial program, MPPPRE, which creates a binary file containing decomposition information for the problem. If MPPPRE is not used, MPP/LS-DYNA3D will do the decomposition at run time. The advantages of using MPPPRE are:

- The parallel machine is not tied up while the decomposition is done.

- The start up time of the parallel run is greatly reduced.

- If the problem is to be run more than once, the decomposition need only be done once.

- The problem can later be run on any number of processors which evenly divides the number of processors for which the decomposition was performed.

Output Files and Post-Processing

For performance reasons, many of the ASCII output files normally created by LS-DYNA3D have been combined into a new binary format used by MPP/LS-DYNA3D. There is a post-processing program DUMPBDB which reads this binary database of files and produces as output the corresponding ASCII files. The new binary files will be created in the directory specified as the global directory in the pfile (See Section 6.) The file (one per processor) are named **DBOUT.***nnnn*, where *nnnn* is replaced by the four digit processor number. To convert these files to ASCII three steps are required, as follows:

```
cd <global directory>
cat DBOUT.* > DBOUT
DUMPBDB ABOUT
```

Many of the normal LS-DYNA3D files will have corresponding collections of files produced by MPP/LS-DYNA3D, with one per processor. These include D3DUMP files (new names = D3DUMP.*nnnn*), the MESSAG files (now MES*nnnn*) and others. Most of these will be found in the local directory specified in the pfile.

The format of the D3PLOT file has not been changed. It will be created in the global directory, and can be directly handled with our graphics post-processor, LS-TAURUS.

Parallel Specific Options

There is a new command line option: **p=pfile**. **pfile** contains MPP specific parameters that affect the execution of the program. The file is split into sections with several options in each section. Currently, three sections: directory, decomposition, and contact, are available. First, here is a sample **pfile:**

```
directory {
global rundir
local /tmp/rundir
}
contact {
inititer 4
}
```

This file is case insensitive and free format input, with the exception that each word or bracket must be surrounded on both sides with either a space, tab, or new line character. The section and options currently supported are:

- **Directory.** Holds directory specific options

  global *path*. Relative path to a directory accessible to all processors. This directory will be created if necessary.
  Default = current working directory

  local *path*. Relative path to a processor specific local directory for scratch files. This directory will be created if necessary. This is of primary use on systems where each processor has its own local disk.
  Default = global path

- **Decomposition.** Holds decomposition specific options.

  file *filename*. The name of the file that holds the decomposition information. This file will be created if the pre-decomposition program is being run. Otherwise, it is expected to exist in the current working directory. If the filename does not end with the extension ".pre" then this extension is added. If this option is not specified, there is no default value.

  numproc *n*. For pre-decomposition only. The problem will be decomposed for n processors. The resulting decomposition file can later be used on any number of processors that evenly divides *n*.

costinc *n*. The elements involved in contact are considered to be this much more computationally expensive during decomposition. The average thin shell is given a weight of about 30, so setting costinc to 30 would indicate that each shell element involved in contact is about twice as computationally expensive as a normal shell element.
Default = 0

method *name*. Currently, there are three decomposition methods supported. Method **rsb** is Recursive Spectral Bisection. This method is only available when using the pre-decomposition program. Method **greedy** is a much faster but less sophisticated method. Method **rcb** is Recursive Coordinate Bisection. The impact on overall runtime is problem dependent, but generally not very large. Other methods are under consideration.
Default = rsb

expdir *n*. This only applies when using Recursive Coordinate Bisection where *n* = 1 specifies the X coordinate direction, 2 the Y and 3 the Z. For a full explanation see the following item.
Default = 1

expsf *t*. This only applies when using Recursive Coordinate Bisection. The model will be compressed by a factor of t in the coordinate direction indicated by the keyword **expdir** before RCB is performed. This in no way affects the geometry of the actual model, but it has the effect of expanding the decomposition domains in the indicated direction by a factor of 1/*t*.. Preliminary experience indicates that this can be used to provide much improved load balance for contact problems. For example, if **expdir** is set to the punch travel direction for a sheet metal stamping problem, and **expsf** is given as 0, each processor will be responsible for a whole column of the problem. This result in the contact work being very equally distributed among the processors, and in some such problems can result in dramatic speed improvements over the other decomposition methods.
Default = 1

show. If this keyword appears in the decomposition section, the D3PLOT file is doctored so that the decomposition can be viewed with the post-processor. Displaying material 1 will show that portion of the problem assigned to processor 0, and so on. The problem will not actually be run, but the code will terminate once the initial D3PLOT state has been written.

- **Contact.** Holds contact specific option. Any conflicting option that might be specified in the problem itself, in accordance with the 920 version manual, are ignored.

  bucket *n*. Specifies the frequency for bucket sort contact searches.
  Default = 200

  ntrack *n*. Specifies the number of contact segments to keep track of per slave node. Increasing this number requires more storage, and will have some impact on speed. For sheet metal stamping problems, values of 1 or 2 are probably adequate, depending on the problem configuration and definition of contact interfaces.
  Default = 3

  maxiter *n*. Specifies the maximum number of Newton iterations while finding the contact point during bucket sort searches. Iteration will continue until the parametric coordinates of contact point change by less than .5% of the segment size, or the maximum number of iterations is achieved.
  Default = 8

  inititer *n*. During contact interface initialization, an attempt is made to move the slave nodes to eliminate initial penetrations. An iterative approach is used, since moving the nodes in one direction may cause problems in a different direction, particularly with the single sided contact options. This parameter specifies the maximum number of iterations to attempt. After the final iteration, any nodes which still have significant penetrations (see "brickoff" below) are deleted from the contact interface. Each processor creates a message file in its local directory which contains among other things, a list of all nodes moved and those nodes deleted during this process. The file name is given by appending a

4 digit processor number to the string "MES" so that for example, the message file from processor 3 is MES0003.
Default = 4

bufnumsf *n*. Sets the number of message buffers during contact equal to the number of the processors involved in the contact surface, divided by *n*. Larger values of *n* result in less memory being used but may negatively impact performance.
Default = 2

bufsizesf *n*. Sets the size of the contact message buffers equal to the maximum possible message size divided by n. Larger values of n will result in messages being split into pieces when sent, saving memory but possibly impacting performance.
Default = 2

bigmem. If this keyword appears, it is equivalent to setting bufnumsf and bufsize both equal to 1. It requires the most memory, but will guarantee the no contact related message ever gets split or ever has to wait for another message to complete before it can be sent. Due to the message passing characteristics of IBM's SP2, and the large memory generally available, bigmem is turned on for this machine by default and the other buffer related keywords are ignored.

Here is a sample pfile with every possible option specified:

```
directory {
 global rundir
 local /tmp/rundir
}

contact {
 bucket 250
 ntrack 4
 maxiter 12
 inititer 6
 bufnumsf 2
 bufsizesf 1
```

```
 bigmem
}

decomposition {
 file dcfile32
 numproc 64
 costinc 10
 method rcb
 show
}
```

Modeling Tips on MPP's

Due to the nature of the parallel contact algorithm, it is more efficient to have as few contact interfaces as possible. For example, a metal forming problem might traditionally be set up with several contact surfaces: blank-die, blank-binder, blank-punch, etc. Speed increases may be obtained by replacing these with a single contact surface with the blank as slave and die, binder, and punch all on the master surfaces. The total amount of contact to be computed may be the same, but the computation might run faster. Similarly for crash applications, it will almost always be faster to create one large type 13 contact region rather than many small contact interfaces.

*AIRBAG

The keyword *AIRBAG provides a way of defining thermodynamic behavior of the gas flow into the airbag as well as a reference configuration for the fully inflated bag. The keyword control cards in this section are defined in alphabetical order:

*AIRBAG_OPTION
*AIRBAG_REFERENCE_GEOMETRY

***AIRBAG_OPTION**

Options include the following thermodynamic relationships:

SIMPLE_PRESSURE_VOLUME
SIMPLE_AIRBAG_MODEL
ADIABATIC_GAS_MODEL
WANG_NEFSKE
WANG_NEFSKE_JETTING
WANG_NEFSKE_MULTIPLE_JETTING
LOAD_CURVE
LINEAR_FLUID

Purpose: Define an airbag or control volume.

*AIRBAG

Card Format

	1	2	3	4	5	6	7	8
Variable	SID	SIDTYP	RBID	VSCA	PSCA	VINI	MWD	SPSF
Type	I	I	I	F	F	F	F	F
Default	none	0	0	1.	1.	0.	0.	0.
Remarks			optional					

VARIABLE	DESCRIPTION
SID	Set ID
SIDTYP	Set type: EQ.0: segment, NE.0: part IDs.
RBID	Rigid body ID for user defined activation subroutine: EQ.0: the control volume is active from time zero, EQ.n: user sensor subroutine flags the start of the inflation. Load curves are offset by initiation time. See Appendix B.
VSCA	Volume scale factor, V_{sca} (default=1.0)
PSCA	Pressure scale factor, P_{sca} (default=1.0)
VINI	Initial filled volume, V_{ini}
MWD	Mass weighted damping factor, D
SPSF	Stagnation pressure scale factor, $0 <= \gamma <= 1$

The first card is necessary for all airbag options. The sequence for the following cards which is different for each option is explained on the next pages.

Lumped parameter control volumes are a mechanism for determining volumes of closed surfaces and applying a pressure based on some thermodynamic relationships. The volume is specified by a list of polygons similar to the pressure boundary condition cards or by specifying a material subset which represents shell elements which form the closed boundary. All polygon normals must be oriented to face outwards from the control volume. If holes are detected, they are assumed to be covered by planar surfaces.

V_{sca} and P_{sca} allow for unit system changes from the inflator to the finite element model. There are two sets of volume and pressure used for each control volume. First, the finite element model computes a volume ($V_{femodel}$) and applies a pressure ($P_{femodel}$). The thermodynamics of a control volume may be computed in a different unit system; thus, there is a separate volume ($V_{cvolume}$) and pressure ($P_{cvolume}$) which are used for integrating the differential equations for the control volume. The conversion is as follows:

$$V_{cvolume} = (V_{sca}V_{femodel})\text{-}V_{ini}$$

$$P_{femodel} = P_{sca}\, P_{cvolume}$$

Damping can be applied to the structure enclosing a control volume by using a mass weighted damping formula:

$$\vec{F}_i = m_i D\left(\vec{V}_i - \vec{V}_{cg}\right)$$

where $\vec{F}_i$ is the damping force, m_i is the nodal mass, $\vec{V}_i$ is the velocity for a node, $\vec{V}_{cg}$ is the mass weighted average velocity of the structure enclosing the control volume, and D is the damping factor.

An alternative, separate damping is based on the stagnation pressure concept. The stagnation pressure is roughly the maximum pressure on a flat plate oriented normal to a steady state flow field. The stagnation pressure is defined as $p = \gamma\rho V^2$ where V is the normal velocity of the control volume relative to the ambient velocity, ρ is the ambient air density, and γ is a factor which varies from 0 to 1 and has to be chosen by the user. Small values are recommended to avoid excessive damping.

Additional cards required for user defined sensor subroutines (See Appendix B), define only if RBID>0.

If the rigid body material number is non-zero then include the following card sets which provide the input parameters for the user defined subroutine. Up to 25 parameters may be used with each control volume.

Card Format

	1	2	3	4	5	6	7	8
Variable	N							
Type	I							
Default	none							

Card Format (Define up to 25 constants for the user subroutine. Input only the number of cards necessary, i.e. for nine constants use 2 cards)

	1	2	3	4	5	6	7	8
Variable	C1	C2	C3	C4	C5			
Type	F	F	F	F	F			
Default	0.	0.	0.	0.	0.			

VARIABLE	DESCRIPTION
N	Number of input parameters (not to exceed 25)
C1,...CN	Up to 25 constants user subroutine.

Additional card required for SIMPLE_PRESSURE_VOLUME option

	1	2	3	4	5	6	7	8
Variable	CN	BETA						
Type	F	F						
Default	none	none						

VARIABLE	DESCRIPTION
CN	Constant
BETA	Scale factor, .β

The relationship is the following:

$$\text{Pressure} = \beta \frac{CN}{Relative\ Volume}$$

$$Relative\ Volume = \frac{Current\ Volume}{Initial\ Volume}$$

The pressure is then a function of the ratio of current volume to the initial volume. The constant, CN, is used to establish a relationship known from the literature. The scale factor β is simply used to scale the given values. This simple model can be used when an initial pressure is given and no leakage, no temperature, and no input mass flow is assumed. A typical application is the modeling of air in automobile tires.

Additional cards required for SIMPLE_AIRBAG_MODEL option

Card 1

	1	2	3	4	5	6	7	8
Variable	CV	CP	T	LCID	MU	A	PE	RO
Type	F	F	F	I	F	F	F	F
Default	none	none	none	none	none	none	none	none

Card 2

Variable	LOU							
Type	I							
Default	0							

VARIABLE	DESCRIPTION
CP	Heat capacity at constant pressure
CV	Heat capacity at constant volume
T	Temperature of input gas
LCID	Load curve ID specifying input mass flow rate. See *DEFINE_CURVE.
MU	Shape factor for exit hole, μ: LT.0.0: $\|\mu\|$ is the load curve number defining the shape factor as a function of absolute pressure.
A	Exit area, A: GE.0.0: A is the exit area and is constant in time, LT.0.0: |A| is the load curve number defining the exit area as a function of absolute pressure.
PE	Ambient pressure, p_e
RO	Ambient density, ρ
LOU	**Optional** load curve ID giving mass flow out versus gauge pressure in bag. See *DEFINE_CURVE.

The gamma law equation of state used to determine the pressure in the airbag:

$$p = (\gamma - 1)\rho e$$

where p is the pressure, ρ is the density, e is the specific internal energy of the gas, and γ is the ratio of the specific heats:

$$\gamma = \frac{c_p}{c_v}$$

From conservation of mass, the time rate of change of mass flowing into the bag is given as:

$$\frac{dM}{dt} = \frac{dM_{in}}{dt} - \frac{dM_{out}}{dt}$$

The inflow mass flow rate is given by the load curve ID, LCID. Leakage, the mass flow rate out of the bag, can be modeled in two alternative ways. One is to give an exit area with the corresponding shape factor, then the load curve ID, LOU, must be set to zero. The other is to define a mass flow out by a load curve, then μ and A have to both be set to zero.

Additional card required for ADIABATIC_GAS_MODEL option

	1	2	3	4	5	6	7	8
Variable	PSF	LCID	GAMMA	P0	PE	RO		
Type	F	I	F	F	F	F		
Default	1.0	none	none	none	none	none		

VARIABLE	DESCRIPTION
PSF	Pressure scale factor
LCID	Optional load curve for preload flag. See *DEFINE_CURVE.
GAMMA	Ratio of specific heats
P0	Initial pressure (gauge)
PE	Ambient pressure
RO	Initial density of gas

The optional load curve ID, LCID, defines a preload flag. During the preload phase the function value of the load curve versus time is zero, and the pressure in the control volume is given as:

$$p = PSF\ p_0$$

When the **first nonzero** function value is encountered, the preload phase stops and the ideal gas law applies for the rest of the analysis. If LCID is zero, no preload is performed.

The gamma law equation of state for the adiabatic expansion of an ideal gas is used to determine the pressure after preload:

$$p = (\gamma - 1)\rho e$$

where p is the pressure, ρ is the density, e is the specific internal energy of the gas, and γ is the ratio of the specific heats:

$$\gamma = \frac{c_p}{c_v}$$

The pressure above is the absolute pressure, the resultant pressure acting on the control volume is:

$$p_s = PSF\left(p - p_e\right)$$

where PSF is the pressure scale factor. Starting from the initial pressure p_0 an initial internal energy is calculated:

$$e_0 = \frac{p_0 + p_e}{\rho(\gamma - 1)}$$

Additional 4 cards are required for all WANG_NEFSKE models

Card 1

Variable	CV	CP	T	LCT	LCMT	TVOL		
Type	F	F	F	I	I	F		
Default	none	none	0.	0	none	0.		

Card 2

Variable	C23	LCC23	A23	LCA23	CP23	LCCP23	AP23	LCAP23
Type	F	I	F	I	F	I	F	I
Default	none	0	none	0	none	0	0.0	0

Card 3

Variable	PE	RO	GC	LCEFR	POVER	PPOP		
Type	F	F	F	I	F	F		
Default	none	none	none	0	0.0	0.0		

If the inflator is modeled, LCMT=0, define, the following card. If not, define but leave blank.

Card 4	1	2	3	4	5	6	7	8
Variable	IOC	IOA	IVOL	IRO	IT	LCBF		
Type	F	F	F	F	F	I		
Default	none	none	none	none	none	none		

VARIABLE	DESCRIPTION
CV	Heat capacity at constant volume
CP	Heat capacity at constant pressure
T	Temperature of input gas. For temperature variations a load curve, LCT, may be defined.
LCT	Optional load curve number defining temperature of input gas versus time. This overides columns T.
LCMT	Load curve specifying input mass flow rate or tank pressure versus time. If the tank volume, TVOL, is nonzero the curve ID is assumed to be tank pressure versus time. If LCMT=0, then the inflator has to be modeled, see Card 6.
TVOL	Tank volume which is required only for the tank pressure versus time curve, LCMT.
C23	Vent orifice coefficient which applies to exit hole. Set to zero if LCC23 is defined below.
LCC23	Load curve number defining the vent orifice coefficient which applies to exit hole as a function of time. A nonzero value for C23 overrides LCC23.
A23	Vent orifice area which applies to exit hole. Set to zero if LCA23 is defined below.
LCA23	Load curve number defining the vent orifice area which applies to exit hole as a function of absolute pressure. A nonzero value for A23 overrides LCA23.
CP23	Orifice coefficient for leakage (fabric porosity). Set to zero if LCCP23 is defined below.

VARIABLE	DESCRIPTION
LCCP23	Load curve number defining the orifice coefficient for leakage (fabric porosity) as a function of time. A nonzero value for CP23 overrides LCCP23.
AP23	Area for leakage (fabric porosity)
LCAP23	Load curve number defining the area for leakage (fabric porosity) as a function of (absolute) pressure. A nonzero value for AP23 overrides LCAP23.
PE	Ambient pressure
RO	Ambient density
GC	Gravitational conversion constant (mandatory - no default). If consistent units are being used for all parameters in the airbag definition then unity should be input.
LCEFR	Optional curve for exit flow rate versus (gauge) pressure
POVER	Initial relative overpressure (gauge), P_{over} in control volume
PPOP	Relative pressure (gauge) for initiating exit flow, P_{pop}
IOC	Inflator orifice coefficient
IOA	Inflator orifice area
IVOL	Inflator volume
IRO	Inflator density
IT	Inflator temperature
LCBF	Load curve defining burn fraction versus time

The gamma law equation of state for the adiabatic expansion of an ideal gas is used to determine the pressure after preload:

$$p = (\gamma - 1)\rho e$$

where p is the pressure, ρ is the density, e is the specific internal energy of the gas, and γ is the ratio of the specific heats:

$$\gamma = \frac{c_p}{c_v}$$

A pressure relation is defined:

$$Q = \frac{p_e}{p_2}$$

where p_e is the external pressure and p_2 is the internal pressure in the bag. A critical pressure relationship is defined as:

$$Q_{crit} = \left(\frac{2}{\gamma + 1} \right)^{\gamma / \gamma - 1}$$

where γ is the ratio of specific heats:

$$\gamma = \frac{c_p}{c_v}$$

If

$$Q \geq Q_{crit} \quad \text{then} \quad Q = Q_{crit}$$

Wang and Nefske define the mass flow through the vents and leakage by

$$\dot{m}_{23} = C_{23} A_{23} \frac{p_2}{R\sqrt{T_2}} Q^{1/\gamma} \sqrt{2 g_c \left(\frac{\gamma R}{\gamma - 1} \right) \left(1 - Q^{\gamma - 1 / \gamma} \right)}$$

and

$$\dot{m}'_{23} = C'_{23} A'_{23} \frac{p_2}{R\sqrt{T_2}} Q^{1/\gamma} \sqrt{2 g_c \left(\frac{\gamma R}{\gamma - 1} \right) \left(1 - Q^{\gamma - 1 / \gamma} \right)}$$

It must be noted that the gravitational conversion constant has to be given in consistent units. As an alternative to computing the mass flow out of the bag by the Wang-Nefske model, a curve for the exit flow rate depending on the internal pressure can be taken. Then, no definitions for C23, LCC23, A23, LCA23, CP23, LCCP23, AP23, and LCAP23 are necessary.

The airbag inflator assumes that the control volume of the inflator is constant and that the amount of propellant reacted can be defined by the user as a tabulated curve of fraction reacted versus time. A pressure relation is defined:

$$Q_{crit} = \frac{p_c}{p_i} = \left(\frac{2}{\gamma + 1} \right)^{\frac{\gamma}{\gamma - 1}}$$

where p_c is a critical pressure at which sonic flow occurs, p_I, is the inflator pressure. The exhaust pressure is given by

$$p_e = p_a \quad if \quad p_a \geq p_c$$
$$p_e = p_c \quad if \quad p_a < p_c$$

where p_a is the pressure in the control volume. The mass flow into the control volume is governed by the equation:

$$\dot{m}_{in} = C_O A_O \sqrt{2 p_I \rho_I} \sqrt{\frac{g_c \gamma \left(Q^{\frac{2}{\gamma}} - Q^{\frac{\gamma+1}{\gamma}} \right)}{\gamma - 1}}$$

where C_O, A_O, and ρ_I are the inflator orifice coefficient, area, and gas density, respectively.

Further additional 2 cards are required for JETTING models

The following additional cards are defined for the WANG_NEFSKE_JETTING and WANG_NEFSKE_MULTIPLE_JETTING options, two further cards are defined for each option. The jet may be defined by specifying either the coordinates of the jet focal point, jet vector head and secondary jet focal point, or by specifying three nodes located at these positions. The nodal point option is recommended when the location of the airbag changes as a function of time.

Define either card below but <u>not</u> both:

1st additional card of 2 required for WANG_NEFSKE_JETTING option

Card 1 1 2 3 4 5 6 7 8

Variable	XJFP	YJFP	ZJFP	XJVH	YJVH	ZJVH	CA	BETA
Type	F	F	F	F	F	F	F	F
Default	none	none	none	none	none	none	none	none
Remark	1	1	1	1	1	1		

1st additional card of 2 required for WANG_NEFSKE_MULTIPLE_JETTING option

Card 1	1	2	3	4	5	6	7	8
Variable	XJFP	YJFP	ZJFP	XJVH	YJVH	ZJVH	LCJRV	BETA
Type	F	F	F	F	F	F	F	F
Default	none	none	none	none	none	none	none	none
Remark	1	1	1	1	1	1		

2nd additional card of 2 required for WANG_NEFSKE_JETTING and WANG_NEFSKE_MULTIPLE_JETTING option

Card 2	1	2	3	4	5	6	7	8
Variable	XSJFP	YSJFP	ZSJFP	PSID	ANGLE	NODE1	NODE2	NODE3
Type	F	F	F	I	F	I	I	I
Default	none	none	none	none	none	0	0	0
Remark						1	1	1

VARIABLE	DESCRIPTION
XJFP	x-coordinate of jet focal point, i.e., the virtual origin in Figure 1.1. See Remark 1 below.
YJFP	y-coordinate of jet focal point, i.e., the virtual origin in Figure 1.1.
ZJFP	z-coordinate of jet focal point, i.e., the virtual origin in Figure 1.1.
XJVH	x-coordinate of jet vector head to defined code centerline
YJVH	y-coordinate of jet vector head to defined code centerline
ZJVH	z-coordinate of jet vector head to defined code centerline

VARIABLE	DESCRIPTION
CA	Cone angle, α, defined in radians.
LCRJV	Load curve ID giving the spatial jet relative velocity distribution, see Figures 1.2 and 1.3. The jet velocity is determined from the inflow mass rate and scaled by the load curve function value corresponding to the value of the angle ψ. Typically, the values on the load curve vary between 0 and unity. See *DEFINE_CURVE.
BETA	Efficiency factor, β, which scales the final value of pressure obtained from Bernoulli's equation.
XSJFP	x-coordinate of secondary jet focal point, passenger side bag. If the coordinate of the secondary point is (0,0,0) then a conical jet (driver's side airbag) is assumed.
YSJFP	y-coordinate of secondary jet focal point
ZSJFP	z-coordinate of secondary jet focal point
PSID	Optional part set ID, see *PART. If zero all elements are included in the airbag.
ANGLE	Cutoff angle in degrees. The relative jet velocity is set to zero for angles greater than the cutoff. See Figure 1.3.
NODE1	Node ID located at the jet focal point, i.e., the virtual origin in Figure 1.1. See Remark 1 below.
NODE2	Node ID for node along the axis of the jet .
NODE3	Optional node ID located at secondary jet focal point.

Remark:

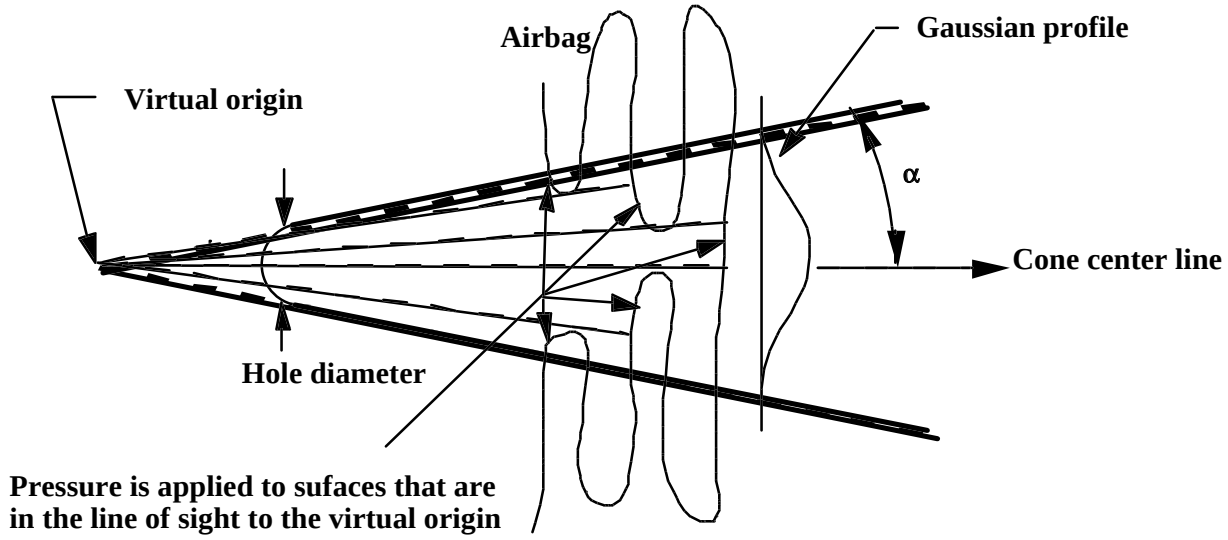
1. It is assumed that the jet direction is defined by the coordinate method (XJFP, YJFP, ZJFP) and (XJVH, YJVH, ZJVH) unless both NODE1 and NODE2 are defined. In which case the coordinates of the nodes give by NODE1, NODE2 and NODE3 will override (XJFP, YJFP, ZJFP) and (XJVH, YJVH, ZJVH). The use of nodes is recommended if the airbag system is undergoing rigid body motion. The nodes should be attached to the vehicle to allow for the coordinates of the jet to be continuously updated with the motion of the vehicle.

The jetting option provides a simple model to simulate the real pressure distribution in the airbag during the breakout and early unfolding phase. Only the sufaces that are in the line of sight to

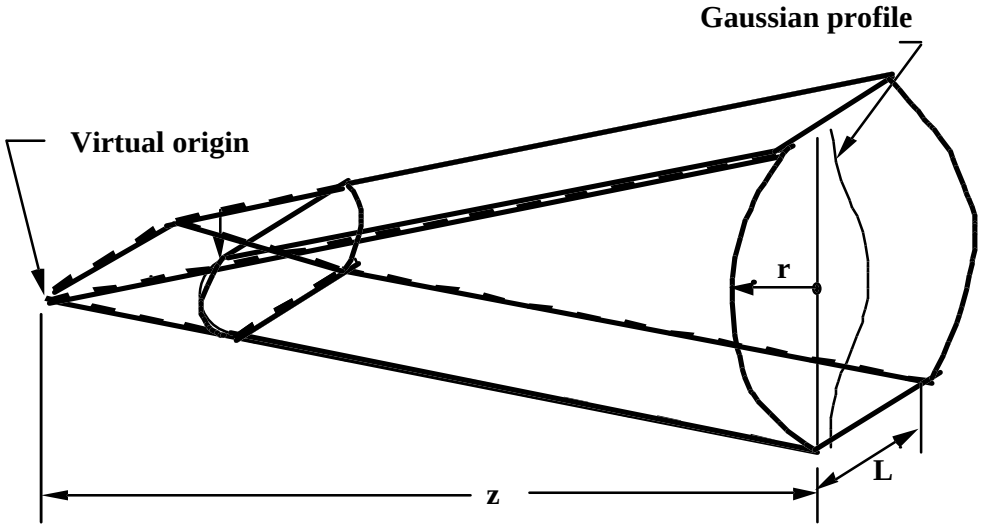
the virtual origin have an increased pressure applied. With the optional load curve LCRJV, the pressure distribution with the code can be scaled according to the so-called relative jet velocity distribution.

For passenger side airbags the cone is replaced by a wedge type shape. The first and secondary jet focal points define the corners of the wedge and the angle α then defines the wedge angle.

Instead of applying pressure to all surfaces in the line of sight of the virtual origin(s), a part set can be defined to which the pressure is applied.



a.



b.

Figure 1.1 Jetting configuration for (a.) driver's side airbag and (b.) the passenger's side bag.

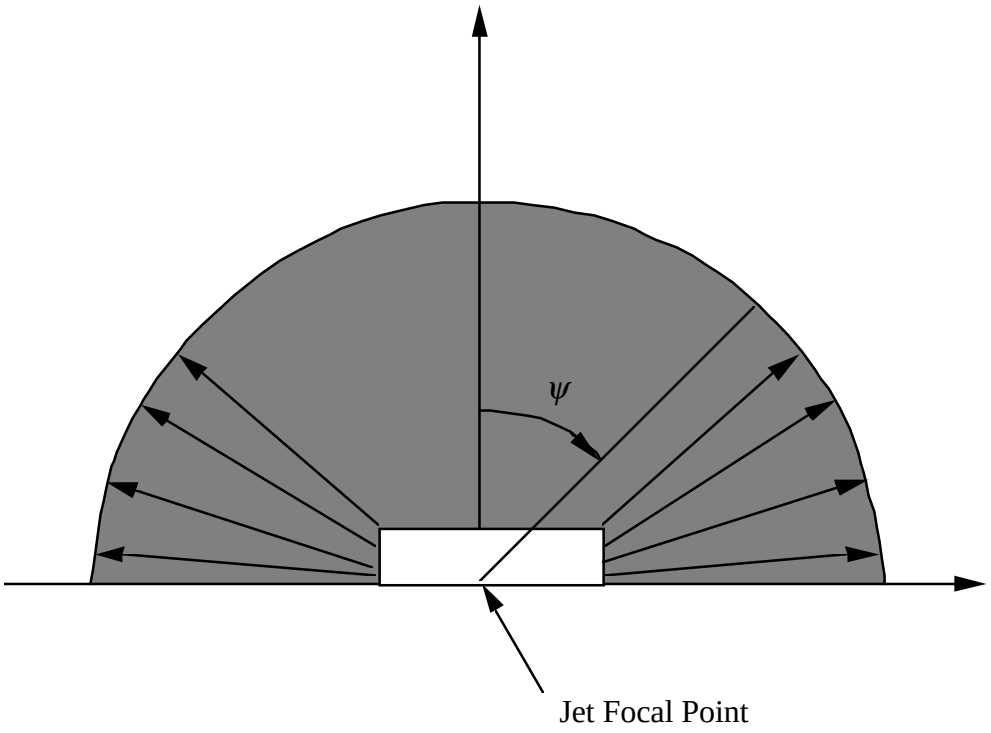


Figure 1.2 Multiple jet model for driver's side airbag.

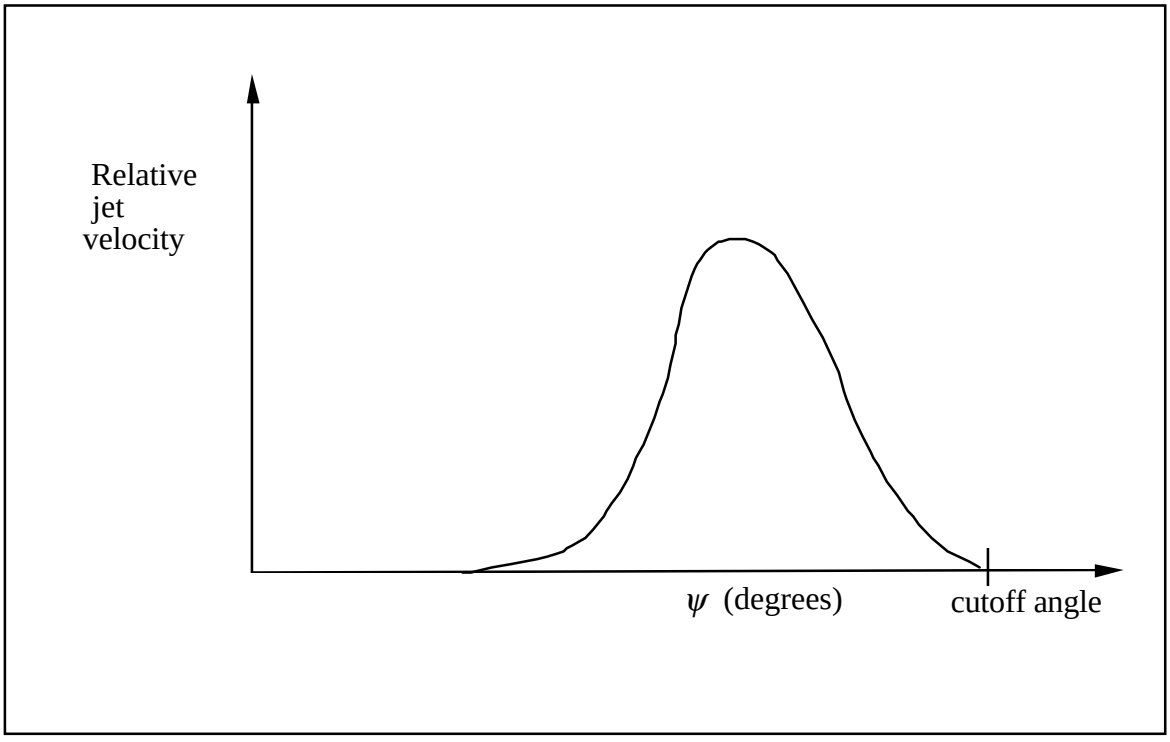


Figure 1.3 Normalized jet velocity versus angle for multiple jet driver's side airbag.

Additional card required for LOAD_CURVE option

	1	2	3	4	5	6	7	8
Variable	STIME	LCID						
Type	F	I						
Default	0.0	none						

VARIABLE	DESCRIPTION
STIME	Time at which pressure is applied. The load curve is offset by this amount.
LCID	Load curve ID defining pressure versus time, see *DEFINE_CURVE.

Within this simple model the control volume is inflated with a pressure defined as a function of time. The pressure is uniform throughout the control volume.

Additional card required for LINEAR_FLUID option

	1	2	3	4	5	6	7	8
Variable	BULK	RO	LCID					
Type	F	F	I					
Default	none	none	none					

VARIABLE	DESCRIPTION
BULK	K, bulk modulus of the fluid in the control volume
RO	ρ, density of the fluid
LCID	F(t), input flow curve defining mass per unit time, see *DEFINE_CURVE.

Pressure is determined from:

$$P(t) = K \ln\left(\frac{V_0(t)}{V(t)}\right)$$

where

$P(t)$ =Pressure,

$V(t)$ =Volume of fluid in compressed state,

$V_0(t)$ $= V_0(t) = \frac{M(t)}{\rho}$ =Volume of fluid in uncompressed state,

$M(t)$ $= M(0) + \int F(t)dt$ =current fluid mass,

$M(0)$ $= V(0)\rho$ =mass of fluid at time zero $P(0) = 0$.

This model is for the simulation of hydroforming processes or similar problems. The pressure is controlled by the mass flowing into the volume and by the current volume. The pressure is uniformly applied to the control volume.

***AIRBAG_INTERACTION**

Purpose: To define two connected airbags which vent into each other.

Define one card for each airbag interaction definition

	1	2	3	4	5	6	7	8
Variable	AB1	AB2	AREA	SF				
Type	I	I	F	F				
Default	none	none	none	none				

VARIABLE	DESCRIPTION
AB1	First airbag ID, as defined on *AIRBAG card.
AB2	Second airbag ID, as defined on *AIRBAG card.
AREA	Orifice area between connected bags. LT.0.0: |AREA| is the load curve ID defining the orifice area as a function of absolute pressure.
SF	Shape factor. LT.0.0: |SF| is the load curve ID defining vent orifice coefficient as a function of relative time.

This input is valid for the following airbag types:

*AIRBAG_SIMPLE_AIRBAG_MODEL
*AIRBAG_WANG_NEFSKE
*AIRBAG_WANG_NEFSKE_JETTING
*AIRBAG_WANG_NEFSKE_MULTIPLE_JETTING

The airbags must contain the same gas, i.e. C_p, C_V and g must be the same. The flow between bags is governed by formulas which are similar to those of Wang-Nefske, except that choked flow is currently ignored. This will be added later.

*AIRBAG_REFERENCE_GEOMETRY

Purpose: If the reference configuration of the airbag is taken as the folded configuration, the geometrical accuracy of the deployed bag will be affected by both the stretching and the compression of elements during the folding process. Such element distortions are very difficult to avoid in a folded bag. By reading in a reference configuration such as the final unstretched configuration of a deployed bag, any distortions in the initial geometry of the folded bag will have no effect on the final geometry of the inflated bag. This is because the stresses depend only on the deformation gradient matrix:

$$F_{ij} = \frac{\partial x_i}{\partial X_j}$$

where the choice of X_j may coincide with the folded or unfold configurations. It is this unfolded configuration which may be specified here.

Card Format (I8,3E16.0)

Card 1	1	2	3	4	5	6	7	8	9	10
Variable	NID	X		Y		Z				
Type	I	F		F		F				
Default	none	0.		0.		0.				
Remarks										

VARIABLE	DESCRIPTION
NID	Node number
X	x coordinate
Y	y coordinate
Z	z coordinate

*ALE

The keyword *ALE provides a way of defining options that are specific to the keyword capability.

***ALE_SMOOTHING**

Purpose: This smoothing constraint keeps a node at its initial parametric location along a line between two other nodes. This constraint is active during each mesh smoothing operation.

Card Format

	1	2	3	4	5	6	7	8
Variable	SNID	MNID1	MNID2					
Type	I	I	I					
Default	none	none	none					

VARIABLE	DESCRIPTION
SNID	Slave node ID, see Figure 2.1.
MNID1	First master node ID
MNID2	Second master node ID

Abritrary Lagrangean Eulerian meshes are defined via the choice of the element type-only solids elements can be used, and the *CONTROL_ALE card.

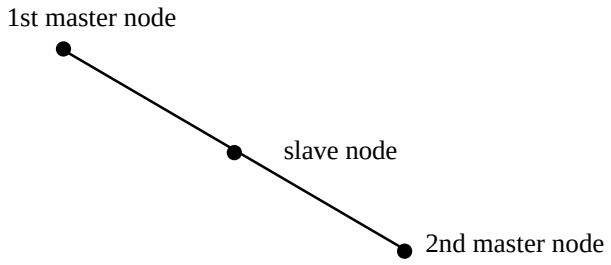


Figure 2.1 This simple constraint, which ensures that a slave node remains on a straight line between two master nodes, is sometimes necessary during ALE smoothing.

*BOUNDARY

The keyword *BOUNDARY provides a way of defining imposed motions on boundary nodes. The keyword control cards in this section are defined in alphabetical order:

*BOUNDARY_CONVECTION_OPTION
*BOUNDARY_CYCLIC
*BOUNDARY_FLUX_OPTION
*BOUNDARY_NON_REFLECTING
*BOUNDARY_PRESCRIBED_MOTION_OPTION
*BOUNDARY_PRESSURE_OUTFLOW_OPTION
*BOUNDARY_RADIATION_OPTION
*BOUNDARY_SLIDING_PLANE
*BOUNDARY_SPC_OPTION
*BOUNDARY_SYMMETRY_FAILURE
*BOUNDARY_TEMPERATURE_OPTION
*BOUNDARY_USA_SURFACE

*BOUNDARY

BOUNDARY_CONVECTION_*OPTION

Available options are:

SEGMENT

SET

Purpose: Define convection boundary conditions for a thermal or coupled thermal/structural analysis. Two cards are defined for each option.

For the **SET** option define the following card:

Card Format (Card 1 of 2)

Card 1	1	2	3	4	5	6	7	8
Variable	SSID							
Type	I							
Default	none							

For the **SEGMENT** option define the following card:

Card Format (Card 1 of 2)

Card 1	1	2	3	4	5	6	7	8
Variable	N1	N2	N3	N4				
Type	I	I	I	I				
Default	none	none	none	none				

Define the following card for both options:

Card Format (Card 2 of 2)

Card 1	1	2	3	4	5	6	7	8
Variable	HLCID	HMULT	TLCID	TMULT				
Type	I	F	I	F				
Default	none	1.0	none	1.0				

VARIABLE	DESCRIPTION
SSID	Segment set ID, see *SET_SEGMENT.
N1,N2...	Node ID's defining segment.
HLCID	Load curve ID for heat transfer coefficient, *h:* GT.0: function versus time, EQ.0: use constant multiplier value, HMULT, LT.0: function versus temperature.
HMULT	Curve multiplier for *h.*
TLCID	Load curve ID for T_∞ versus time, see *DEFINE_CURVE: EQ.0: use constant multiplier value, TMULT.
TMULT	Curve multiplier for T_∞

A convection boundary condition is calculated using $\dot{q}'' = h(T - T_\infty)$

where

h	heat transfer coefficient
$(T - T_\infty)$	temperature potential

Three alternatives are possible for the heat transfer coefficient which can be a function of time, a function of temperature, or constant. Also, the temperature of the boundary T_∞ can be either constant or a function of time. For both curves, multipliers can be used to scale the values.

*BOUNDARY_CYCLIC

Purpose: Define nodes in boundary planes for cyclic symmetry.

These boundary conditions can be used to model a segment of an object that has rotational symmetry such as an impeller, i.e., Figure 3.1. The segment boundarys, denoted as a side 1 and side 2, may be curved or planar. In this section, a paired list of points are defined on the sides that are to be joined.

Card Format

	1	2	3	4	5	6	7	8
Variable	XC	YC	ZC	NSID1	NSID2			
Type	F	F	F	I	I			
Default	none	none	none	none	none			

VARIABLE	DESCRIPTION
XC	x-component axis vector of axis of rotation
YC	y-component axis vector of axis of rotation
ZC	z-component axis vector of axis of rotation
NSID1	Node set ID for first boundary plane (side 1, see Figure 3.1).
NSID2	Node set ID for second boundary plane (side 2, see Figure 3.1). Each boundary node in this boundary plane is constrained to its corresponding node in the first node set. Node sets NSID1 and NSID2 must contain the same number of nodal points. Care has to be taken that the nodes in both node sets have a location which, if given in cylindrical coordinates, differ all by the same angle.

Only globally defined axes of rotation are possible.

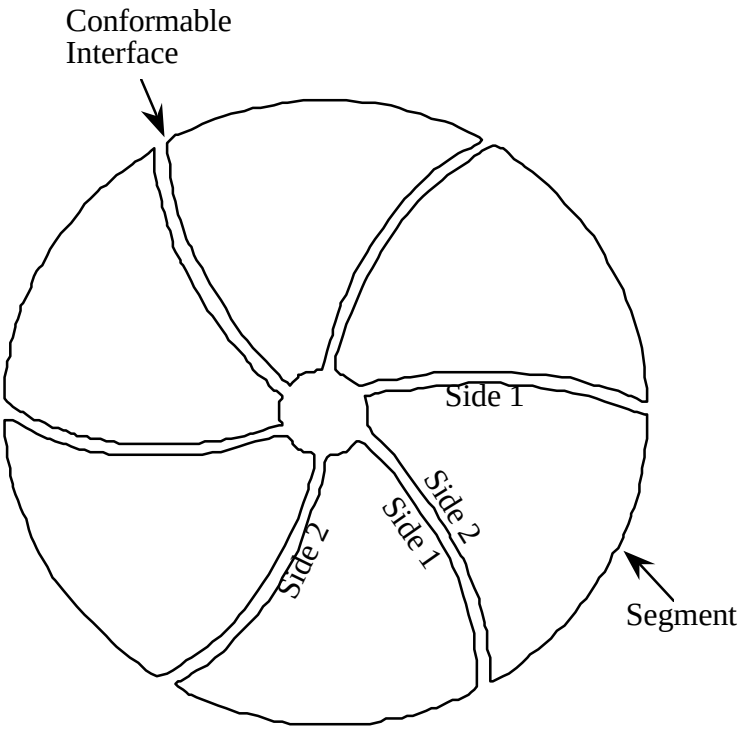


Figure 3.1 With cyclic symmetry only one segment is modeled.

BOUNDARY_FLUX_*OPTION

Available options are:

SEGMENT
SET

Purpose: Define flux boundary conditions for a thermal or coupled thermal/structural analysis. Two cards are defined for each option.

For the **SET** option define the following card:

Card Format (Card 1 of 2)

Card 1	1	2	3	4	5	6	7	8
Variable	SSID							
Type	I							
Default	none							

For the **SEGMENT** option define the following card:

Card Format (Card 1 of 2)

Card 1	1	2	3	4	5	6	7	8
Variable	N1	N2	N3	N4				
Type	I	I	I	I				
Default	none	none	none	none				

Define the following card for both options:

Card Format (Card 2 of 2)

Card 1	1	2	3	4	5	6	7	8
Variable	LCID	MLC1	MLC2	MLC3	MLC4			
Type	I	F	F	F	F			
Default	none	1.0	1.0	1.0	1.0			

VARIABLE	DESCRIPTION
SSID	Segment set ID, see *SET_SEGMENT
N1,N2...	Node ID's defining segment
LCID	Load curve ID for heat flux, see *DEFINE_CURVE: GT.0: function versus time, EQ.0: use constant multiplier values at nodes, LT.0: function versus temperature.
MLC1	Curve multiplier at node N_1, see Figure 3.2.
MLC2	Curve multiplier at node N_2, see Figure 3.2.
MLC3	Curve multiplier at node N_3, see Figure 3.2.
MLC4	Curve multiplier at node N_4, see Figure 3.2.

Three definitions for heat flux are possible. Heat flux can be a function of time, a function of temperature, or constant values that are maintained throughout the calculation. With the definition of multipliers at each node of the segment, a bilinear spatial variation can be assumed.

By convention, heat flow is positive in the direction of the surface outward normal vector. Surface definition is in accordance with the right hand rule. The outward normal vector points to the right as one progresses from node N_1-N_2-N_3-N_4. See Figure 3.2.

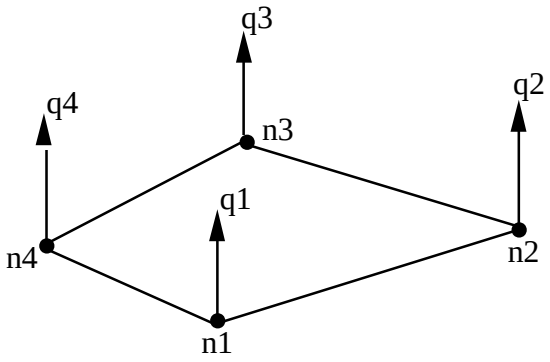


Figure 3.2. Nodal number determines outward normal.

***BOUNDARY_NON_REFLECTING**

Purpose: Define a non-reflecting boundary. This option applies to continuum domains modeled with solid elements, as indefinite domains are usually not modelled.

Card Format

	1	2	3	4	5	6	7	8
Variable	SSID	AD	AS					
Type	I	F	F					
Default	none	0.0	0.0					

VARIABLE	DESCRIPTION
SSID	Segment set ID, see *SET_SEGMENT.
AD	Default activation flag for dilatational waves. (on.eq.0.0, off.ne.0.0)
AS	Default activation flag for shear waves. (on.eq.0.0, off.ne.0.0)

With the two optional switches, the influence of reflecting waves can be studied. For geomechanical problems it becomes an option to limit the size of the models.

*BOUNDARY

*BOUNDARY_PRESCRIBED_MOTION_OPTION

Available options include:

NODE
SET
RIGID

Purpose: Define an imposed nodal motion (velocity, acceleration, or displacement) on a node or a set of nodes. Also velocities and displacements can be imposed on rigid bodies.

Card Format

Card 1

1	2	3	4	5	6	7	8
Variable	*type*ID	DOF	VAD	LCID	SF	VID	DEATH	
Type	I	I	I	I	F	I	F	
Default	none	none	0	none	1.	0	0.	

Card is required if DOF=9,10,11 on the first card. If DOF<9, skip this card.

Card 2

1	2	3	4	5	6	7	8
Variable	OFFSET1	OFFSET2						
Type	F	F						
Default	0.	0.						

VARIABLE	DESCRIPTION
NID, NSID, PID	Node ID (NID), nodal set ID (NSID), SEE *SET_NODE, or part ID (PID), see *PART, for a rigid body.
DOF	Applicable degrees-of-freedom: EQ. 1: x-translational degree-of-freedom, EQ. 2: y-translational degree-of-freedom, EQ. 3: z-translational degree-of-freedom, EQ. 4: translational motion in direction given by the VID. Movement on plane normal to the vector is permitted. EQ.-4: translational motion in direction given by the VID. Movement on plane normal to the vector is not permitted. This option does not apply to rigid bodies. EQ.5: x-rotational degree-of-freedom, EQ. 6: y-rotational degree-of-freedom, EQ. 7: z-rotational degree-of-freedom, EQ. 8: rotational motion about the vector given by the VID. Rotation about the normal axes is permitted. EQ.-8: rotational motion about the vector given by the VID. Rotation about the normal axes is not permitted. This option does not apply to rigid bodies. EQ. 9: y/z degrees-of-freedom for node rotating about the x-axis at location (OFFSET1,OFFSET2) in the yz-plane, *point (y,z)*, EQ.10: z/x degrees-of-freedom for node rotating about the y-axis at location (OFFSET1,OFFSET2) in the zx-plane, *point (z,x)*, EQ.11: x/y degrees-of-freedom for node rotating about the z-axis at location (OFFSET1,OFFSET2) in the xy-plane, *point (x,y)*.
VAD	Velocity/Acceleration/Displacement flag: EQ.0: velocity (rigid bodies and nodes), EQ.1: acceleration (nodes only), EQ.2: displacement (rigid bodies and nodes).
LCID	Load curve ID to describe motion value versus time, see *DEFINE_CURVE.
SF	Load Curve Scale Factor
VID	Vector ID for DOF values of 4 or 8, see *DEFINE_VECTOR.
DEATH	Time imposed motion/constraint is removed: EQ.0.0: default set to 10^{28}.
OFFSET1	Offset for DOF types 9-11 (y, z, x direction)
OFFSET2	Offset for DOF types 9-11 (z, x, y direction)

Abitrary translations and rotations are possible. Rotations around local axis can be defined either by setting DOF=8 or by using the offset option of DOF>8. The load curve scale factor can be used for simple modifications or unit adjustments.

BOUNDARY_PRESSURE_OUTFLOW_*OPTION

Available options are
SEGMENT
SET

Purpose: Define pressure outflow boundary conditions. These bounday conditions are attached to solid elements using the eulerian ambient formulation (7) and defined to be pressure outflow ambient elements (3). See *SECTION_SOLID_OPTION.

For the SET option define the following card

Card Format

Card 1	1	2	3	4	5	6	7	8
Variable	SSID							
Type	I							
Default	none							

For the SEGMENT option define the following card

Card Format

Card 1	1	2	3	4	5	6	7	8
Variable	N1	N2	N3	N4				
Type	I	I	I	I				
Default	none	none	none	none				

VARIABLE	DESCRIPTION
SSID	Segment set ID
N1,N2...	Node ID’s defining segment

BOUNDARY_RADIATION_*OPTION

Available options are:

SEGMENT
SET

Purpose: Define radiation boundary conditions for a thermal or coupled thermal/structural analysis. Two cards are defined for each option.

For the **SET** option define the following card:

Card Format (Card 1 of 2)

Card 1	1	2	3	4	5	6	7	8
Variable	SSID							
Type	I							
Default	none							

For the **SEGMENT** option define the following card:

Card Format (Card 1 of 2)

Card 1	1	2	3	4	5	6	7	8
Variable	N1	N2	N3	N4				
Type	I	I	I	I				
Default	none	none	none	none				

Define the following card for both options:

Card Format (Card 2 of 2)

	1	2	3	4	5	6	7	8
Variable	RFLCID	RFMULT	TILCID	TIMULT				
Type	I	F	I	F				
Default	none	1.0	none	1.0				

VARIABLE	DESCRIPTION
SSID	Segment set ID, see *SET_SEGMENT
N1,N2...	Node ID's defining segment
RFLCID	Load curve ID for radiation factor *f*, see *DEFINE_CURVE: GT.0: function versus time, EQ.0: use constant multiplier value, FMULT, LT.0: function versus temperature.
RFMULT	Curve multiplier for *f*, see *DEFINE_CURVE
TILCID	Load curve ID for $T\infty$ versus time, see *DEFINE_CURVE: EQ.0: use constant multiplier, TIMULT.
TIMULT	Curve multiplier for $T\infty$

A radiation boundary condition is calculated using a radiant-heat-transfer coefficient. Set $\dot{q}'' = h_r (T - T\infty)$, where h_r is a radiant-heat-transfer coefficient defined as

$$h_r = f\,(T + T_\infty)(T^2 + T^2{}_\infty)$$

The exchange factor, *F*, is a characterization of the effect of the system geometry, emissivity and reflectivity on the capability of radiative transport between surfaces. The radiation boundary condition data cards require specification of the product, $f = F\sigma$, and $T\infty$ for the boundary surface.

*BOUNDARY_SLIDING_PLANE

Purpose: Define a sliding symmetry plane. This option applies to continuum domains modeled with solid elements.

Card Format

	1	2	3	4	5	6	7	8
Variable	NSID	VX	VY	VZ	COPT			
Type	I	F	F	F	I			
Default	none	0	0	0	0			

VARIABLE	DESCRIPTION
NSID	Nodal set ID, see *SET_NODE
VX	x-coordinate of vector defining normal or vector
VY	y-coordinate of vector defining normal or vector
VZ	z-coordinate of vector defining normal or vector
COPT	Option: EQ.0: node moves on normal plane, EQ.1: node moves only in vector direction.

Any node may be constrained to move on an arbitrarily oriented plane or line depending on the choice of COPT. Each boundary condition card defines a vector originating at (0,0,0) and terminating at the coordinates defined above. Since an arbitrary magnitude is assumed for this vector, the specified coordinates are non-unique and define only a direction.

***BOUNDARY_SPC_OPTION**

Available options include:

NODE
SET

Purpose: Define nodal single point constraints.

Card Format

	1	2	3	4	5	6	7	8
Variable	NID/NSID	CID	DOFX	DOFY	DOFZ	DOFRX	DOFRY	DOFRZ
Type	I	I	I	I	I	I	I	I
Default	none	0	0	0	0	0	0	0

VARIABLE	DESCRIPTION
NID/NSID	Node ID or nodal set ID, see *SET_NODE.
CID	Coordinate system ID, see *DEFINE_COORDINATE_SYSTEM.
DOFX	Insert 1 for translational constraint in local x-direction.
DOFY	Insert 1 for translational constraint in local y-direction.
DOFZ	Insert 1 for translational constraint in local z-direction.
DOFRX	Insert 1 for rotational constraint about local x-axis.
DOFRY	Insert 1 for rotational constraint about local y-axis.
DOFRZ	Insert 1 for rotational constraint about local z-axis.

Constraints are applied if a value of 1 is given for DOFxx. A value of zero means no constraint.

*BOUNDARY_SYMMETRY_FAILURE

Purpose: Define a symmetry plane with a failure criterion. This option applies to continuum domains modeled with solid elements.

Card Format

	1	2	3	4	5	6	7	8
Variable	SSID	FS	VTX	VTY	VTZ	VHX	VHY	VHZ
Type	I	F	F	F	F	F	F	F
Default	none	0	0	0	0	0	0	0

VARIABLE	DESCRIPTION
SSID	Segment set ID, see *SET_SEGMENT
FS	Tensile failure stress > 0.0. The average stress in the elements surrounding the boundary nodes in a direction perpendicular to the boundary is used.
VTX	x-coordinate of tail of a normal vector originating on the wall (tail) and terminating in the body (head) (i.e., vector points from the symmetry plane into the body).
VTY	y-coordinate of tail
VTZ	z-coordinate of tail
VHX	x-coordinate of head
VHY	y-coordinate of head
VHZ	z-coordinate of head

A plane of symmetry is assumed for the nodes on the boundary at the tail of the vector given above. Only the motion perpendicular to the symmetry plane is constrained. After failure the nodes are set free.

BOUNDARY_TEMPERATURE_*OPTION

Available options are:

NODE
SET

Purpose: Define temperature boundary conditions for a thermal or coupled thermal/structural analysis.

Card Format

	1	2	3	4	5	6	7	8
Variable	NID/SID	LCID	CMULT					
Type	I	I	F					
Default	none	0	1.0					

VARIABLE	DESCRIPTION
NID/SID	Node ID/Node Set ID, see *SET_NODE_OPTION
LCID	Load curve ID for temperature versus time: EQ.0: use the constant multiplier value given below by CMULT.
CMULT	Curve multiplier for temperature

If no load curve ID is given, then a constant boundary temperature is assumed. CMULT is also used to scale the load curve values.

*BOUNDARY_USA_SURFACE

Purpose: Define a surface for coupling with the USA boundary element code [DeRuntz, 1993]. The outward normal vectors should point into the fluid media.

Card Format

	1	2	3	4	5	6	7	8
Variable	SSID	WETDRY	NBEAM					
Type	I	I	I					
Default	none	0	0					

VARIABLE	DESCRIPTION
SSID	Segment set ID, see *SET_SEGMENT
WETDRY	Wet surface flag: EQ.0: dry, no coupling, EQ.1: wet, coupled with USA.
NBEAM	Number of beam elements. This is the number defined in the USA input file and is required for LS-DYNA3D solely for the purpose of memory allocation. It is assumed that each beam that is input into USA has a corresponding beam in the LS-DYNA3D input file.

The 4-node surface segment normals must point into the fluid. The total number of beams is then summed over all cards that are input. If beams are defined in USA, NBEAM should be nonzero only on one card in this section.

When running a coupled problem with USA the procedure involves several steps. First, LS-DYNA3D is executed to create a linking file "dyna.pre" used by USA and a dump file "d3dump". The execution lines are:

LS-DYNA3D > outputfilename0 <cr>
i=inputfilename <cr>

Where we note that no prompt is provided for the second line of the input and that <cr> means that the **c**arriage **r**eturn key should be pressed. Then, it is necessary to create the fluid mass matrix by running the code FLUMAS:

FLUMAS < flumasinputfilename > flumasoutputfilename

The ouput file from the LS-DYNA3D run, dyna.pre, is referenced in the input file to FLUMAS. Next, the code AUGMAT which initializes constants and arrays for the staggered solution procedure for the transient analysis is executed:

AUGMAT < augmatinputfilename > augmatoutputfilename

Finally, the coupled solution can begin by again executing LS-DYNA3D:

LS-DYNA3D > outputfilename <cr>
r=d3dump <cr>
*add usainputfilename <cr>

We note that no prompts are provide for the second and third lines of input. The input files, flumasinputfilename, augmatinputfilename, and usainputfilename, are prepared in accordance with the USA code documentation.

It is advisable when running coupled problems to check the ASCII output files to ensure that each run completed normally.

*CONSTRAINED

The keyword *CONSTRAINED provides a way of constraining degrees of freedom to move together in some way. The keyword control cards in this section are defined in alphabetical order:

*CONSTRAINED_EXTRA_NODES_OPTION
*CONSTRAINED_GENERALIZED_WELD_OPTION
*CONSTRAINED_JOINT_OPTION
*CONSTRAINED_JOINT_STIFFNESS_OPTION
*CONSTRAINED_LINEAR
*CONSTRAINED_NODAL_RIGID_BODY_{OPTION}
*CONSTRAINED_NODE_SET
*CONSTRAINED_RIGID_BODIES
*CONSTRAINED_RIGID_BODY_STOPPERS
*CONSTRAINED_RIVET
*CONSTRAINED_SHELL_TO_SOLID
*CONSTRAINED_SHELL_IN_SOLID
*CONSTRAINED_SPOTWELD
*CONSTRAINED_TIE-BREAK
*CONSTRAINED_TIED_NODES_FAILURE

*CONSTRAINED

*CONSTRAINED_EXTRA_NODES_OPTION

Available options include:

NODE
SET

Purpose: Define extra nodes for rigid body.

Card Format

	1	2	3	4	5	6	7	8
Variable	PID	NID/NSID						
Type	I	I						
Default	none	none						

VARIABLE	DESCRIPTION
PID	Part ID of rigid body to which the nodes will be added, see *PART.
NID/NSID	Node (option: _NODE) or node set ID (option: _SET), see *SET_NODE, of added nodes.

This option allows the definition of additional nodes with extra masses, loads or anything else to be constrained to a rigid body such as joints. The extra nodes can be defined at any location, and are assumed to be part of the rigid body. The coordinates of the extra nodes are updated according to the rigid body motion.

***CONSTRAINED_GENERALIZED_WELD_OPTION**

Then the following options are available:

SPOT
FILLET
BUTT

Purpose: Define spot and fillet welds. Coincident nodes are permitted.

Card Format

	1	2	3	4	5	6	7	8
Variable	NSID	CID						
Type	I	I						
Default	none	none						

Additional Card required for the SPOT option:

Card 2	1	2	3	4	5	6	7	8
Variable	TFAIL	EPSF	SN	SS	N	M		
Type	F	F	F	F	F	F		

Additional Card required for the FILLET option:

Card 2	1	2	3	4	5	6	7	8
Variable	TFAIL	EPSF	SIGY	BETA	L	W	A	ALPHA
Type	F	F	F	F	F	F	F	F

*CONSTRAINED

Additional Card required for the BUTT option:

Card 2	1	2	3	4	5	6	7	8
Variable	TFAIL	EPSF	SIGY	BETA	L	D	LT	
Type	F	F	F	F	F	F	F	

VARIABLE	DESCRIPTION
NSID	Nodal set ID, see *SET_NODE_OPTION.
CID	Coordinate system ID for output of data in local system, see *DEFINE_COORDINATE_OPTION.
TFAIL	Failure time for constraint set, t_f. (default=1.E+20)
EPSF	Effective plastic strain at failure, ε^p_{fail} defines ductile failure.
SN	S_n, normal force at failure, only for the brittle failure of spotwelds.
SS	S_s, shear force at failure, only for the brittle failure of spotwelds.
N	n, exponent for normal force, only for the brittle failure of spotwelds.
M	m, exponent for shear force, only for the brittle failure of spotwelds.
SIGY	σ_f, stress at failure for brittle failure.
BETA	β, failure parameter for brittle failure.
L	L, length of fillet/butt weld (see Figure 4.2 and 4.3).
W	w, width of flange (see Figure 4.2).
A	a, width of fillet weld (see Figure 4.2).
ALPHA	α, weld angle (see Figure 4.2) in degrees.
D	d, thickness of butt weld (see Figure 4.3).
LT	L_t, transverse length of butt weld (see Figure 4.3).

Failures can include both the plastic and brittle failures. These can be used either independently or together. Failure occurs when either criteria is met.

Spotweld failure due to plastic straining occurs when the effective nodal plastic strain exceeds the input value, ε^{p}_{fail}. This option can model the tearing out of a spotweld from the sheet metal since the plasticity is in the material that surrounds the spotweld, not the spotweld itself. A least squares algorithm is used to generate the nodal values of plastic strains at the nodes from the element integration point values. The plastic strain is integrated through the element and the average value is projected to the nodes via a least square fit. This option should only be used for the material models related to metallic plasticity and can result in slightly increased run times.

Brittle failure of the spotwelds occurs when:

$$\left(\frac{\max(f_n,0)}{S_n}\right)^n + \left(\frac{|f_s|}{S_s}\right)^m \geq 1$$

where f_n and f_s are the normal and shear interface force. Component f_n contributes for tensile values only. When the failure time, t_f, is reached the nodal rigid body becomes inactive and the constrained nodes may move freely. In Figure 4.1 the ordering of the nodes is shown for the 2 node and 3 node spotwelds. This order is with respect to the local coordinate system where the local z axis determines the tensile direction. The nodes in the spotweld may coincide. The failure of the 3 node spotweld may occur gradually with first one node failing and later the second node may fail. For n noded spotwelds the failure is progressive starting with the outer nodes (1 and n) and then moving inward to nodes 2 and n-1. Progressive failure is necessary to preclude failures that would create new rigid bodies.

Ductile fillet weld failure, due to plastic straining, is treated identically to spotweld failure.

Brittle failure of the fillet welds occurs when:

$$\beta\sqrt{\sigma_n^2 + 3\left(\tau_n^2 + \tau_t^2\right)} \geq \sigma_f$$

where

σ_n = normal stress
τ_n = shear stress in direction of weld (local y)
τ_t = shear stress normal to weld (local x)
σ_f = failure stress
β = failure parameter

Component σ_n is nonzero for tensile values only. When the failure time, t_f, is reached the nodal rigid body becomes inactive and the constrained nodes may move freely. In Figure 4.2 the ordering of the nodes is shown for the 2 node and 3 node fillet welds. This order is with respect to

the local coordinate system where the local z axis determines the tensile direction. The nodes in the fillet weld may coincide. The failure of the 3 node fillet weld may occur gradually with first one node failing and later the second node may fail.

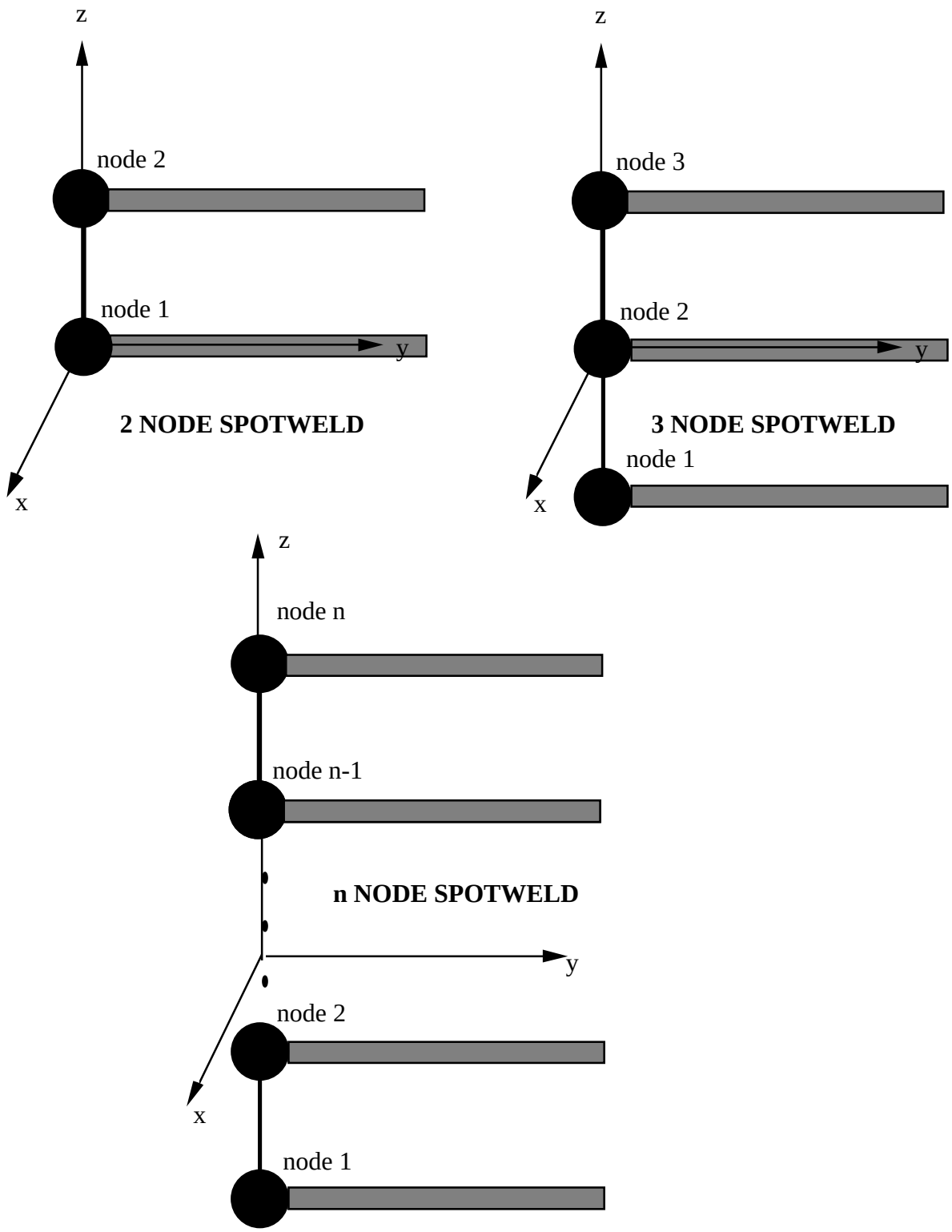


Figure 4.1. Nodal ordering and orientation of the local coordinate system is important for determining spotweld failure.

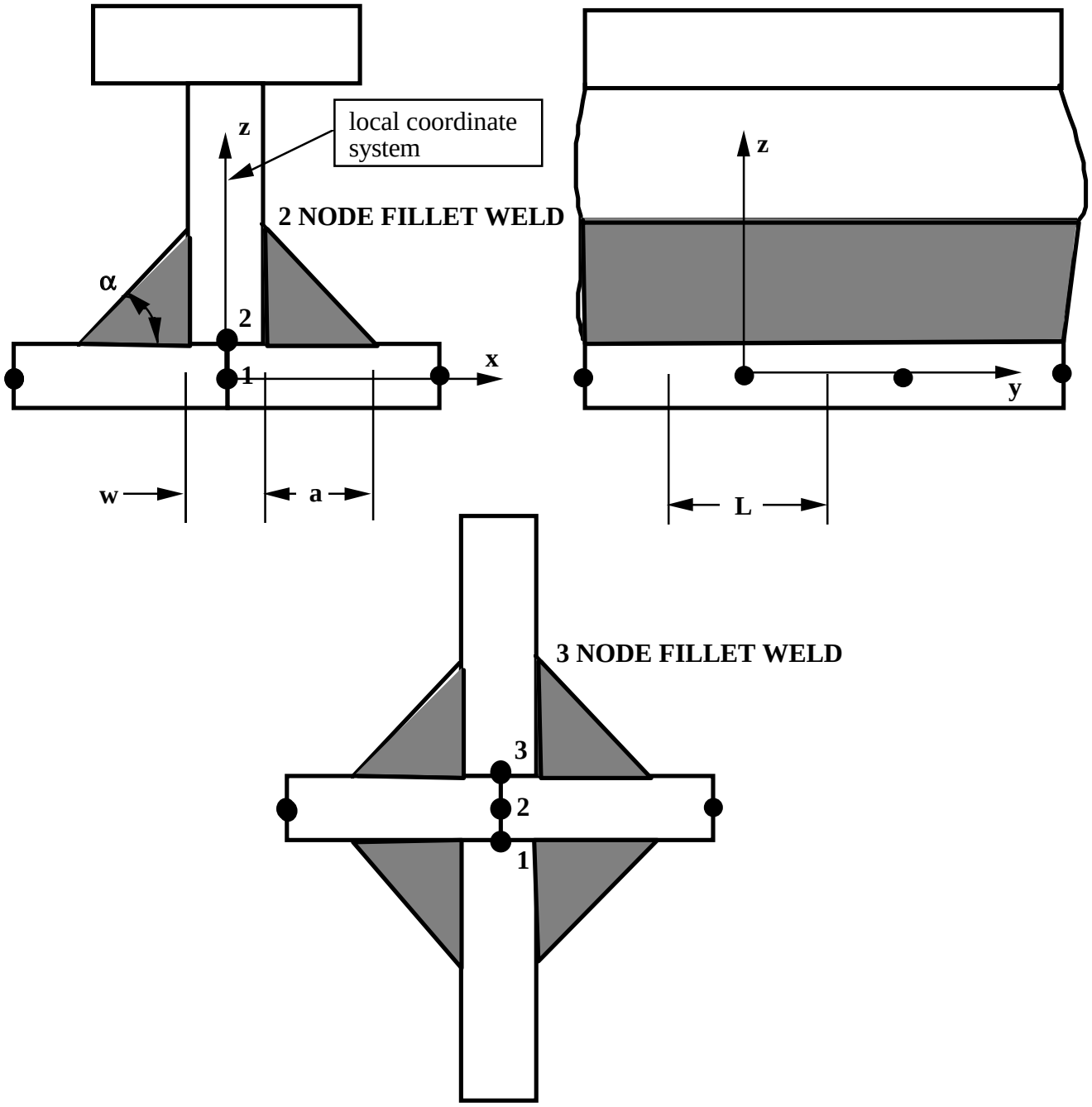


Figure 4.2. Nodal ordering and orientation of the local coordinate system is shown for fillet weld failure. The angle is defined in degrees.

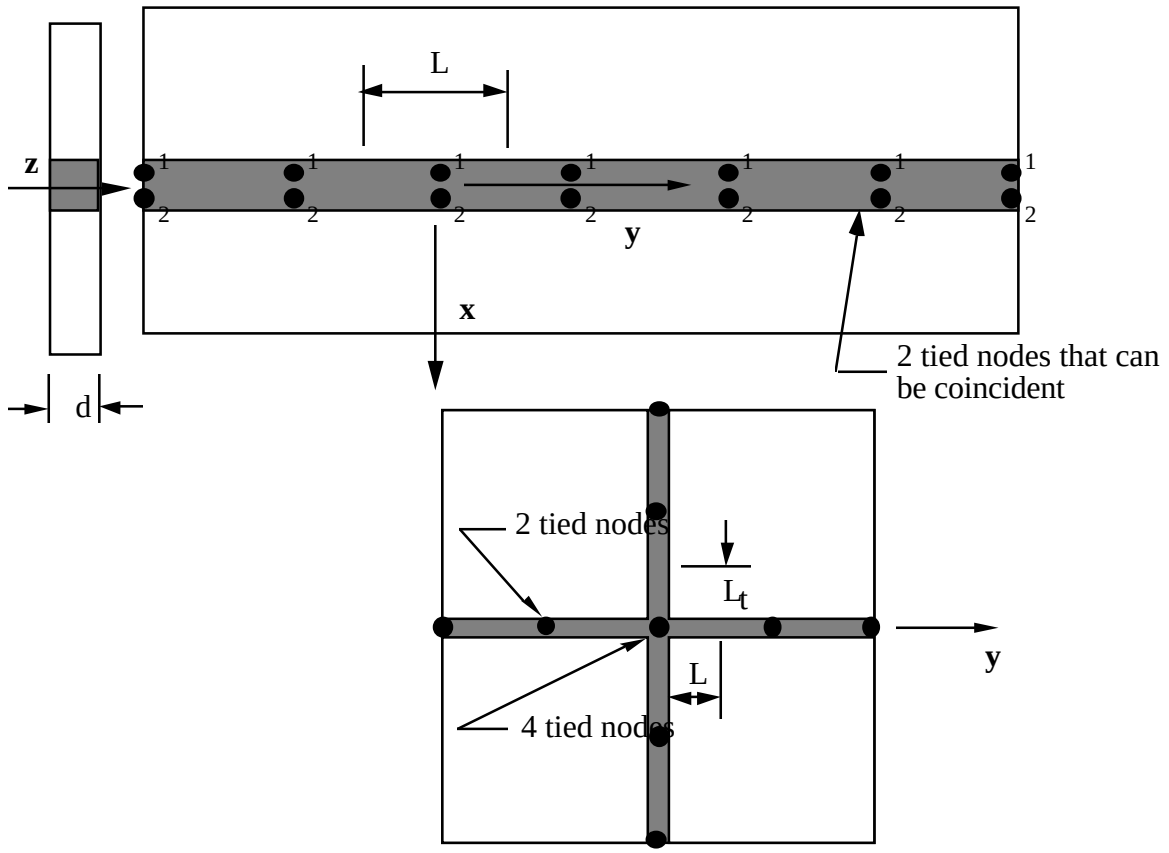


Figure 4.3. Orientation of the local coordinate system and nodal ordering is shown for butt weld failure.

*CONSTRAINED

***CONSTRAINED_JOINT_OPTION**

Options include:

SPHERICAL
REVOLUTE
CYLINDRICAL
PLANAR
UNIVERSAL
TRANSLATIONAL

Purpose: Define a joint between two rigid bodies, see Figure 4.4.

Card Format

Card 1

1	2	3	4	5	6	7	8
N1	N2	N3	N4	N5	N6	RPS	DAMP
I	I	I	I	I	I	F	F
0	0	0	0	0	0	1.0	1.0

VARIABLE	DESCRIPTION
N1	Node 1, in rigid body A. Define for all joint types.
N2	Node 2, in rigid body B. Define for all joint types.
N3	Node 3, in rigid body A. Define for all joint types except SPHERICAL
N4	Node 4, in rigid body B. Define for all joint types except SPHERICAL.
N5	Node 5, in rigid body A. Define only for TRANSLATIONAL joints.
N6	Node 6, in rigid body B. Define only for TRANSLATIONAL joints.
RPS	Relative penalty stiffness (default = 1.0).
DAMP	Damping scale factor on default damping value. (Revolute and Spherical Joints): EQ.0.0: default is set to 1.0, LE.0.01 and GT.0.0: no damping is used.

With one exception, the nodal points within the nodal pairs (1,2), (3,4), and (5,6) (see Figure 4.4) should coincide in the initial configuration, and the nodal pairs should be as far apart as possible to obtain the best behavior. For the Universal Joint the nodes within the nodal pair (3,4) do not coincide, but the lines drawn between nodes (1,3) and (2,4) must be perpendicular.

The geometry of joints is defined in Figure 4.4. Insofar as the penalty method is used, at each time step, the relative penalty stiffness is multiplied by a function dependent on the step size to give the maximum stiffness that will not destroy the stability of the solution. Instabilities can result in the explicit time integration scheme if the penalty stiffness is too large. If instabilities occur, the recommended way to eliminate these problems is to decrease the time step.

For cylindrical joints, by setting node 3 to zero, it is possible to use a cylindrical joint to join a node that is not on a rigid body (node 1) to a rigid body (nodes 2 and 4).

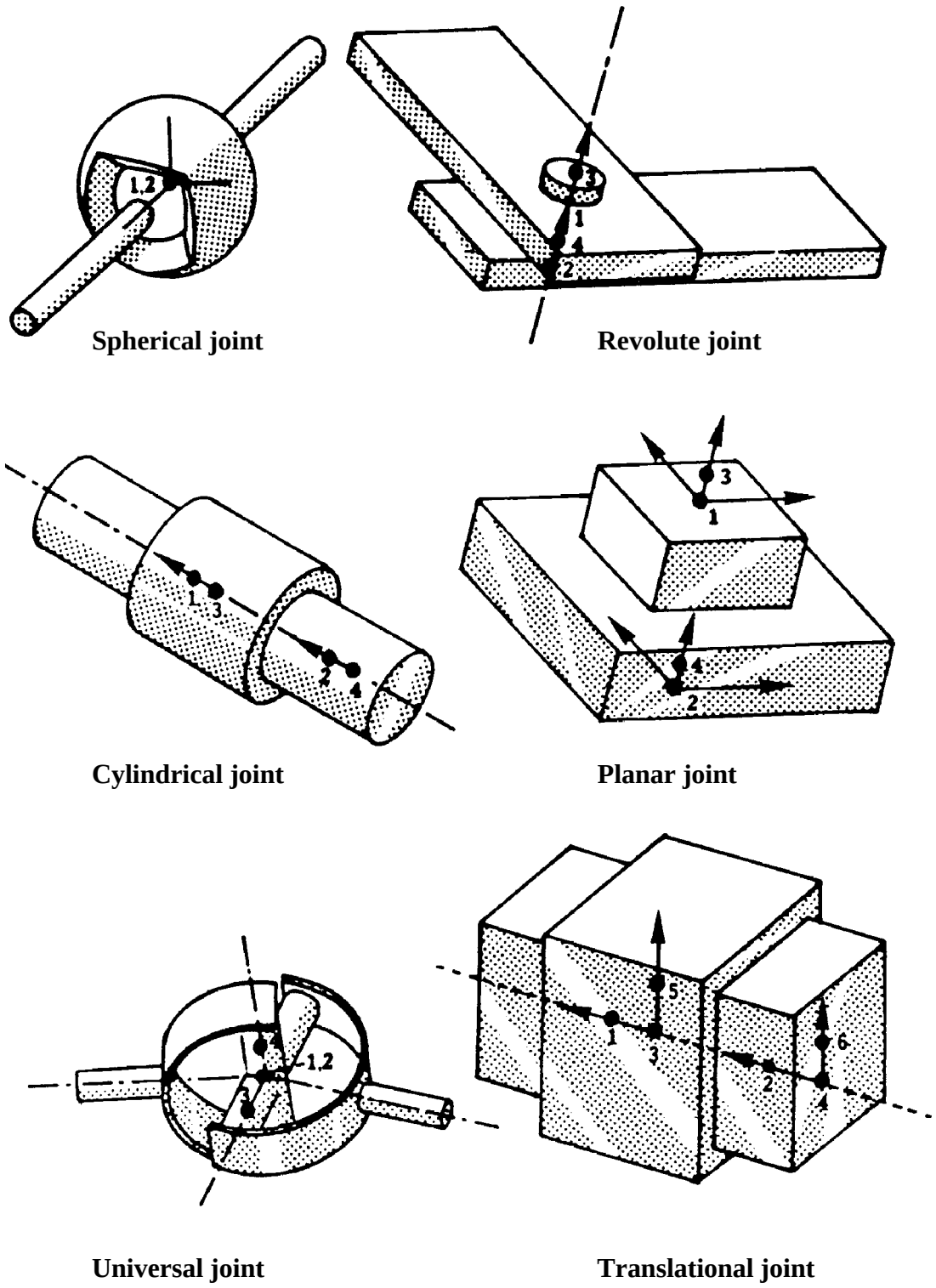


Figure 4.4. Joint definitions.

***CONSTRAINED_JOINT_STIFFNESS_OPTION**

Options include:

GENERALIZED
FLEXION-TORSION

Purpose: Define joint stiffness for joints defined by *CONSTRAINED_JOINT_OPTION.

Card Format (for both options).

Card 1	1	2	3	4	5			
Variable	JSID	PIDA	PIDB	CIDA	CIDB			
Type	I	I	I	I	I			
Default	none	none	none	none	CIDA			

VARIABLE	DESCRIPTION
JSID	Joint stiffness ID
PIDA	Part ID for rigid body A, see *PART.
PIDB	Part ID for rigid body B, see *PART.
CIDA	Coordinate ID for rigid body A, see *DEFINE_COORDINATE_OPTION.
CIDB	Coordinate ID for rigid body B. If zero, the coordinate ID for rigid body A is used, see *DEFINE_COORDINATE_OPTION.

Card Format (2 of 4) (Required for GENERALIZED stiffness)

Card 2	1	2	3	4	5	6		
Variable	LCIDPH	LCIDT	LCIDPS	DLCIDPH	DLCIDT	DLCIDPS		
Type	I	I	I	I	I	I		
Default	none	none	none	none	none	none		

Card Format (3 of 4) (Required for GENERALIZED stiffness)

Card 3	1	2	3	4	5	6		
Variable	ESPH	FMPH	EST	FMT	ESPS	FMPS		
Type	F	F	F	F	F	F		
Default	0.0	0.0	0.0	0.0	0.0	0.0		

Card Format (4 of 4) (Required for GENERALIZED stiffness)

Card 4	1	2	3	4	5	6		
Variable	NSAPH	PSAPH	NSAT	PSAT	NSAPS	PSAPS		
Type	F	F	F	F	F	F		
Default	not used	not used	not used	not used	not used	not used		

VARIABLE	DESCRIPTION
LCIDPH	Load curve ID for ϕ–moment versus rotation in radians. See Figure 4.5. If zero, the applied moment is set to 0.0. See *DEFINE_CURVE.
LCIDT	Load curve ID for θ–moment versus rotation in radians. If zero, the applied moment is set to 0.0. See *DEFINE_CURVE.
LCIDPS	Load curve ID for ψ–moment versus rotation in radians. If zero, the applied moment is set to 0.0. See *DEFINE_CURVE.
DLCIDPH	Load curve ID for ϕ–damping moment versus rate of rotation in radians per unit time. If zero, damping is not considered. See *DEFINE_CURVE.
DLCIDT	Load curve ID for θ–damping moment versus rate of rotation in radians per unit time. If zero, damping is not considered. See *DEFINE_CURVE.
DLCIDPS	Load curve ID for ψ–damping torque versus rate of rotation in radians per unit time. If zero, damping is not considered. See *DEFINE_CURVE.
ESPH	Elastic stiffness per unit radian for friction and stop angles for ϕ rotation. See Figure 4.6. If zero, friction and stop angles are inactive for ϕ rotation.
FMPH	Frictional moment limiting value for ϕ rotation. If zero, friction is inactive for ϕ rotation. This option may also be thought of as an elastic-plastic spring. If a negative value is input then the absolute value is taken as the load curve ID defining the yield moment versus ϕ rotation. See Figure 4.6.
EST	Elastic stiffness per unit radian for friction and stop angles for θ rotation. See Figure 4.6. If zero, friction and stop angles are inactive for θ rotation.
FMT	Frictional moment limiting value for θ rotation. If zero, friction is inactive for θ rotation. This option may also be thought of as an elastic-plastic spring. If a negative value is input then the absolute value is taken as the load curve ID defining the yield moment versus θ rotation. See Figure 4.6.
ESPS	Elastic stiffness per unit radian for friction and stop angles for ψ rotation. See Figure 4.6. If zero, friction and stop angles are inactive for ψ rotation..
FMPS	Frictional moment limiting value for ψ rotation. If zero, friction is inactive for ψ rotation. This option may also be thought of as an elastic-plastic spring. If a negative value is input then the absolute value is taken as the load curve ID defining the yield moment versus ψ rotation. See Figure 4.6.

VARIABLE	DESCRIPTION
NSAPH	Stop angle in degrees for negative ϕ rotation. Ignored if zero.
PSAPH	Stop angle in degrees for positive ϕ rotation. Ignored if zero.
NSAT	Stop angle in degrees for negative θ rotation. Ignored if zero.
PSAT	Stop angle in degrees for positive θ rotation. Ignored if zero.
NSAPS	Stop angle in degrees for negative ψ rotation. Ignored if zero.
PSAPS	Stop angle in degrees for positive ψ rotation. Ignored if zero.

After the stop angles are reached the torques increase linearly to resist further angular motion using the stiffness values on Card 3. Reasonable stiffness values have to be chosen. If the stiffness values are too low or zero, the stop will be violated.

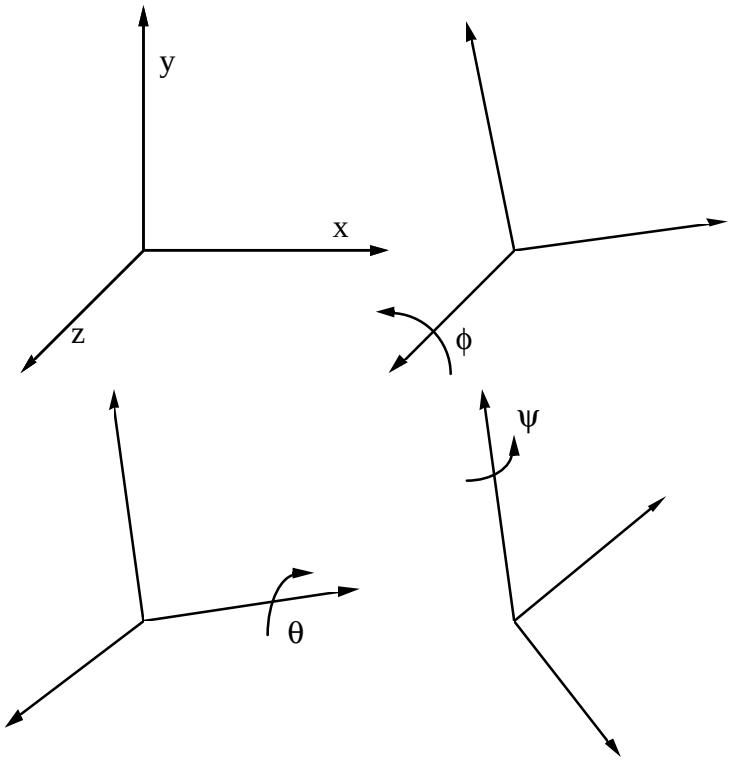


Figure 4.5. Definition of angles for the generalized joint stiffness. The magnitude of the angular rotations are limited by the stop angles defined on Card 4. If the initial local coordinate axes do not coincide, the angles, ϕ, θ, and ψ, will be initialized and torques will develop instantaneously based on the defined load curves.

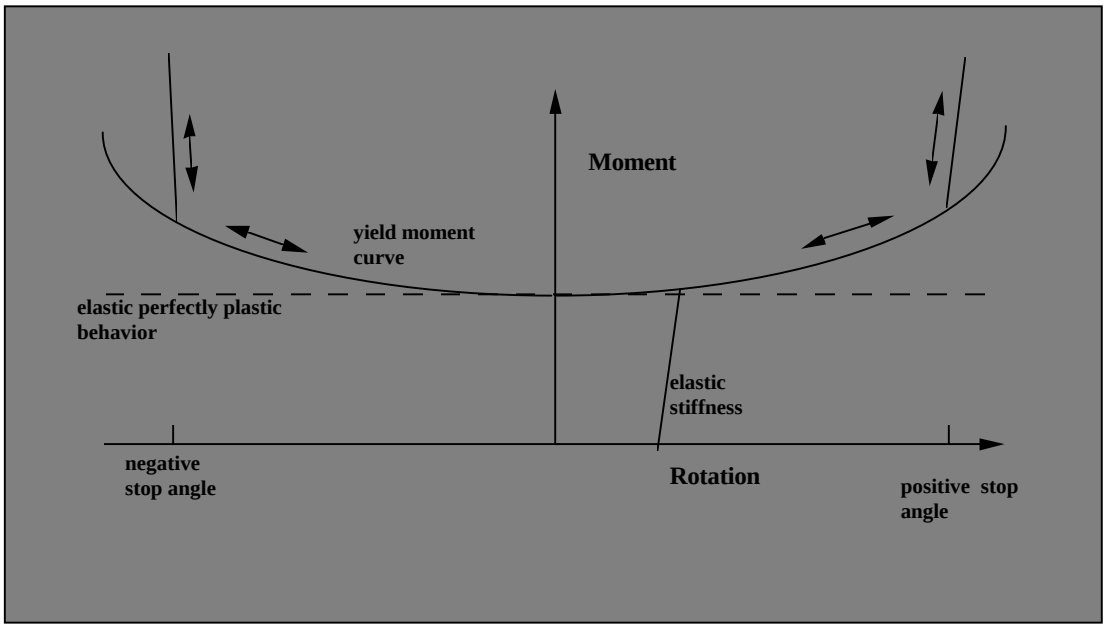


Figure 4.6. Frictional behavior is modeled by a plasticity model. Elastic behavior is obtained once the stop angles are reached. The same elastic stiffness is used to simulate sticking situations.

Card Format (2 of 4) (Required for FLEXION-TORSION stiffness)

Card 2 1 2 3 4 5 6

Variable	LCIDAL	LCIDG	LCIDBT	DLCIDAL	DLCIDG	DLCIDBT		
Type	I	I	I	I	I	I		
Default	none	1.0	none	none	1.0	none		

Card Format (3 of 4) (Required for FLEXION-TORSION stiffness)

Card 3 1 2 3 4

Variable	ESAL	FMAL	ESBT	FMBT				
Type	F	F	F	F				
Default	0.0	0.0	0.0	0.0				

Card Format (4 of 4) (Required for FLEXION-TORSION stiffness)

Card 4 1 2 3

Variable	SAAL	NSABT	PSABT					
Type	F	F	F					
Default	not used	not used	not used					

VARIABLE	DESCRIPTION
LCIDAL	Load curve ID for α–moment versus rotation in radians. See Figure 4.7 where it should be noted that $0 \le \alpha \le \pi$. If zero, the applied moment is set to zero. See *DEFINE_CURVE.
LCIDG	Load curve ID for γ versus a scale factor which scales the bending moment due to the α rotation. This load curve should be defined in the interval $-\pi \le \gamma \le \pi$. If zero the scale factor defaults to 1.0. See *DEFINE_ CURVE.
LCIDBT	Load curve ID for β–torsion moment versus twist in radians. If zero the applied twist is set to zero. See *DEFINE_CURVE.
DLCIDAL	Load curve ID for α–damping moment versus rate of rotation in radians per unit time. If zero, damping is not considered. See *DEFINE_CURVE.
DLCIDG	Load curve ID for γ–damping scale factor versus rate of rotation in radians per unit time. This scale factor scales the α–damping moment. If zero, the scale factor defaults to one. See *DEFINE_CURVE.
DLCIDBT	Load curve ID for β–damping torque versus rate of twist. If zero damping is not considered. See *DEFINE_CURVE.
ESAL	Elastic stiffness per unit radian for friction and stop angles for α rotation, see Figure 4.7. If zero, friction and stop angles are inactive for α rotation..
FMAL	Frictional moment limiting value for α rotation. If zero, friction is inactive for α rotation. This option may also be thought of as an elastic-plastic spring. If a negative value is input then the absolute value is taken as the load curve ID defining the yield moment versus α rotation, see Figure 4.7.
ESBT	Elastic stiffness per unit radian for friction and stop angles for β twist, see Figure 4.7. If zero, friction and stop angles are inactive for β twist.
FMBT	Frictional moment limiting value for β twist. If zero, friction is inactive for β twist. This option may also be thought of as an elastic-plastic spring. If a negative value is input then the absolute value is taken as the load curve ID defining the yield moment versus β rotation, see Figure 4.7.
SAAL	Stop angle in degrees for α rotation where $0 \le \alpha \le \pi$. Ignored if zero.
NSABT	Stop angle in degrees for negative β rotation. Ignored if zero.
PSABT	Stop angle in degrees for positive β rotation. Ignored if zero.

This option simulates a flexion-torsion behavior of a joint in a slightly different fashion than with the generalized joint option.

After the stop angles are reached the torques increase linearly to resist further angular motion using the stiffness values on Card 3. If the stiffness value is too low or zero, the stop will be violated.

The moment resultants generated from the moment versus rotation curve, damping moment versus rate-of-rotation curve, and friction are evaluated independently and are added together.

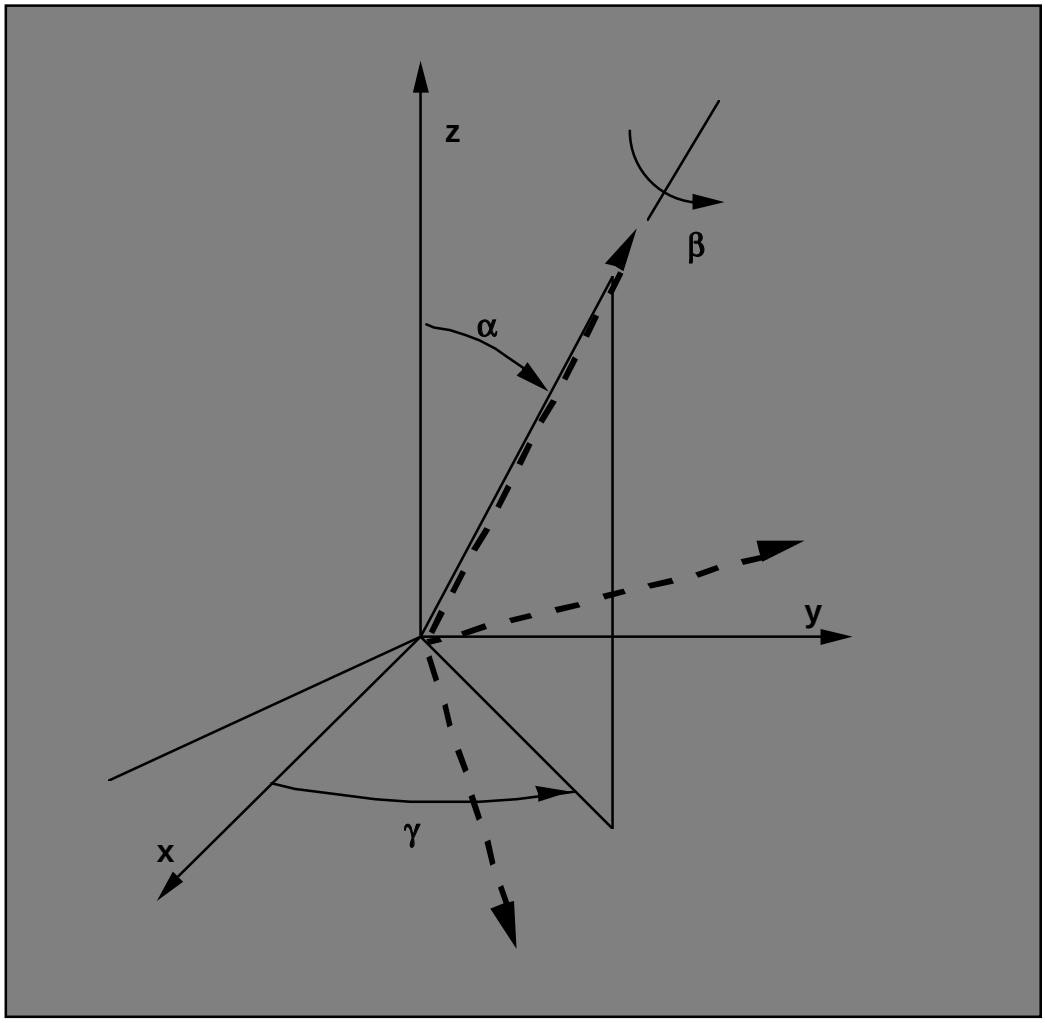


Figure 4.7. Flexion-torsion joint angles. If the initial positions of the local coordinate axes of the two rigid bodies connected by the joint do not coincide, the angles, α and γ, are initialized and torques will develop instantaneously based on the defined load curves. The angle β is also initialized but no torque will develop about the local axis on which β is measured. Rather, β will be measured relative to the computed offset.

*CONSTRAINED

*CONSTRAINED_LINEAR

Purpose: Define linear constraint equations between displacements/rotations, which can be defined in local/global coordinate systems.

Card Formats

Card 1

Variable	NUM							
Type	I							
Default	none							

Define NUM cards below, one card for each nodal point.

Card 2

Variable	NID	DOFX	DOFY	DOFZ	DOFRX	DOFRY	DOFRZ	COEF
Type	I	I	I	I	I	I	I	F
Default	none	0	0	0	0	0	0	0
Remark	1							

VARIABLE	DESCRIPTION
NUM	Number of nodes in equation
NID	Node ID
DOFX	Insert 1 (0) for (no) translational constraint in local x-direction.
DOFY	Insert 1 (0) for (no) translational constraint in local y-direction.

VARIABLE	DESCRIPTION
DOFZ	Insert 1 (0) for (no) translational constraint in local z-direction.
DOFRX	Insert 1 (0) for (no) rotational constraint about local x-axis.
DOFRY	Insert 1 (0) for (no) rotational constraint about local y-axis.
DOFRZ	Insert 1 (0) for (no) rotational constraint about local z-axis.
COEF	Nonzero coefficient, C_k

In this section linear constraint equations of the form:

$$\sum_{k=1}^{n} C_k u_k = C_0$$

can be defined, where u_k are the displacements and C_k are user defined coefficients. Unless LS-DYNA3D is initialized by linking to an implicit code to satisfy this equation at the beginning of the calculation, the constant C_0 is assumed to be zero. The first constrained degree-of-freedom is eliminated from the equations-of-motion:

$$u_1 = C_0 - \sum_{k=2}^{n} \frac{C_k}{C_1} u_k$$

Its velocities and accelerations are given by

$$\dot{u}_1 = -\sum_{k=2}^{n} \frac{C_k}{C_1} \dot{u}_k$$

$$\ddot{u}_1 = -\sum_{k=2}^{n} \frac{C_k}{C_1} \ddot{u}_k ,$$

respectively. In the implementation a transformation matrix, $\underset{\sim}{L}$, is constructed relating the unconstrained, $\underset{\sim}{u}$, and constrained, $\underset{\sim c}{u}$, degrees-of-freedom. The constrained accelerations used in the above equation are given by:

$$\underset{\sim c}{\ddot{u}} = \left[\underset{\sim}{L}^t \underset{\sim}{M} \underset{\sim}{L} \right]^{-1} \underset{\sim}{L}^t \underset{\sim}{F}$$

where $\underset{\sim}{M}$ is the Diagonal lumped mass matrix and $\underset{\sim}{F}$ is the right hand side force vector. This requires the inversion of the condensed mass matrix which is equal in size to the number of constrained degrees-of-freedom minus one.

Remark:

1. Nodes of a nodal constraint equation cannot be members of another constraint equation or constraint set that constrain the same degrees-of-freedom, a tied interface, or a rigid body; i.e. nodes cannot be subjected to multiple, independent, and possibly conflicting constraints. Also care must be taken to ensure that single point constraints applied to nodes in a constraint equation do not conflict with the constraint sets constrained degrees-of-freedom.

***CONSTRAINED_NODAL_RIGID_BODY_OPTION**

If the inertial properties are defined rather than computed, then the following option is available:

INERTIA

Purpose: Define a nodal rigid body. This is a rigid body which consists of the defined nodes. If the INERTIA option is not used, then the inertia tensor is computed from the nodal masses. Arbitrary motion of this rigid body is allowed. If the INERTIA option is used, constant translational and rotational velocities can be defined in a global or local coordinate system.

Card Format

	1	2	3	4	5	6	7	8
Variable	NSID	CID						
Type	I	I						
Default	none	none						

Additional Cards are required for the INERTIA option.

Card 2	1	2	3	4	5	6	7	8
Variable	XC	YC	ZC	TM	IRCS			
Type	F	F	F	F	I			
Default	0	0	0	0	0			

Card 3	1	2	3	4	5	6	7	8
Variable	IXX	IXY	IXZ	IYY	IYZ	IZZ		
Type	F	F	F	F	F	F		
Default	none	0	0	none	0	0		

Card 4	1	2	3	4	5	6	7	8
Variable	VTX	VTY	VTZ	VRX	VRY	VRZ		
Type	F	F	F	F	F	F		
Default	0	0	0	0	0	0		

Optional card required for the IRCS=1.

Card 5	1	2	3	4	5	6	7	8
Variable	XL	YL	ZL	XLIP	YLIP	ZLIP		
Type	F	F	F	F	F	F		
Default	none	none	none	none	none	none		

VARIABLE	DESCRIPTION
NSID	Nodal set ID, see *SET_NODE_OPTION. This nodal set defines the rigid body.
CID	Coordinate system ID for output of data in local system, see *DEFINE_COORDINATE_OPTION. Only necessary if no local system is defined below.
XC	x-coordinate of center of mass
YC	y-coordinate of center of mass
ZC	z-coordinate of center of mass
TM	Translational mass
IRCS	Flag for inertia tensor reference coordinate system: EQ.0: global inertia tensor, EQ.1: principal moments of inertias with orientation vectors as given below.
IXX	I_{xx}, xx component of inertia tensor
IXY	I_{xy} (set to zero if IRCS=1)
IXZ	I_{xz} (set to zero if IRCS=1)
IYY	I_{yy}, yy component of inertia tensor
IYZ	I_{yz} (set to zero if IRCS=1)
IZZ	I_{zz}, zz component of inertia tensor
VTX	x-rigid body translational velocity in global coordinate system.
VTY	y-rigid body translational velocity in global coordinate system.
VTZ	z-rigid body translational velocity in global coordinate system.
VRX	x-rigid body rotational velocity in global coordinate system.
VRY	y-rigid body rotational velocity in global coordinate system.
VRZ	z-rigid body rotational velocity in global coordinate system.

VARIABLE	DESCRIPTION
XL	x-coordinate of local x-axis. Origin lies at (0,0,0).
YL	y-coordinate of local x-axis
ZL	z-coordinate of local x-axis
XLIP	x-coordinate of local in-plane vector
YLIP	y-coordinate of local in-plane vector
ZLIP	z-coordinate of local in-plane vector

Unlike the *CONSTRAINED_NODE_SET which permits only translational motion, here the equations of rigid body dynamics are used to update the motion of the nodes and therefore rotation of the nodal sets is admissible. Mass properties are determined from the nodal masses and coordinates. Inertial properties are defined if and only if the INERTIA option is specified.

The local coordinate system is set up in the following way. After the local x-axis is defined, the local z-axis is computed from the cross-product of the local x-axis vector with the given in-plane vector. Finally, the local y-axis is determined from the cross-product of the local z-axis with the local x-axis.

***CONSTRAINED_NODE_SET**

Purpose: Define nodal constraint sets for translational motion in global coordinates. No rotational coupling. See Figure 4.8.

Card Format

	1	2	3	4	5	6	7	8
Variable	NSID	DOF						
Type	I	I						
Default	none	none						
Remarks								

VARIABLE	DESCRIPTION
NSID	Nodal set ID, see *SET_NODE_OPTION.
DOF	Applicable degrees-of-freedom: EQ. 1: x-translational degree-of-freedom, EQ. 2: y-translational degree-of-freedom, EQ. 3: z-translational degree-of-freedom, EQ. 4: x and y-translational degrees-of-freedom, EQ. 5: y and z-translational degrees-of-freedom, EQ. 6: z and x-translational degrees-of-freedom, EQ. 7: x, y, and z-translational degrees-of-freedom.

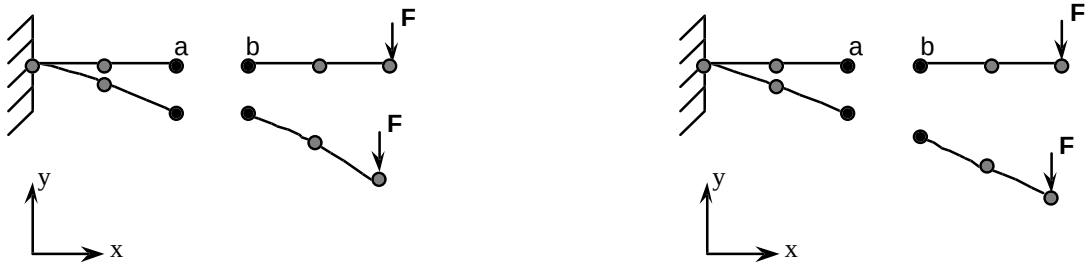
The masses of the nodes are summed up to total mass of the constrained set. It must be noted that the definiton of a nodal rigid body is not possible with this card. For nodal rigid bodies the *CONSTRAINED_NODAL_RIGID_BODY_OPTION has to be used instead.

***CONSTRAINED_NODE_SET**

Since no rotation is permitted, this option should not be used to model rigid body behavior that involves rotations.

***CONSTRAINED_NODAL_RIGID_BO**
***CONSTRAINED_SPOTWELD _**

Behavior is like a rigid beam. These opt may be used to model spotwelds.



Offset nodes a and b are constrained to move together.

Figure 4.8. *CONSTRAINED_NODE_SET can lead to nonphysical responses.

***CONSTRAINED_RIGID_BODIES**

Purpose: Merge two rigid bodies. One rigid body, called slave rigid body, is merged to the other one called a master rigid body.

Card Format

	1	2	3	4	5	6	7	8
Variable	PIDM	PIDS						
Type	I	I						
Default	none	none						

VARIABLE	DESCRIPTION
PIDM	Master rigid body part ID, see *PART.
PIDS	Slave rigid body part ID, see *PART.

All actions valid for the master rigid body, e.g., constraints, given velocity, are now also valid for the newly-created rigid body.

*CONSTRAINED_RIGID_BODY_STOPPERS

Purpose: Rigid body stoppers provide a convenient way of controlling the motion of rigid tooling in metalforming applications. The motion of a "master" rigid body is limited by load curves. This option will stop the motion based on a time dependent constraint. The stopper overrides prescribed velocity and displacement boundary conditions for both the master and slaved rigid bodies. See Figure 4.9.

Card Format

Card 1

Variable	PID	LCMAX	LCMIN	PSIDMX	PSIDMN	LCVMNX	DIR	VID
Type	I	I	I	I	I	I	I	I
Default	required	0	0	0	0	0	required	0

Card 2

Variable	TB	TD						
Type	F	F						
Default	0	10^{21}						

VARIABLE	DESCRIPTION
PID	Part ID of master rigid body, see *PART.
LCMAX	Load curve ID defining the maximum coordinate as a function of time. See *DEFINE_CURVE: EQ.0: no limitation of the maximum displacement.
LCMIN	Load curve ID defining the minimum coordinate as a function of time. See *DEFINE_CURVE: EQ.0: no limitation of the minimum displacement.

VARIABLE	DESCRIPTION
PSIDMX	Optional part set ID of rigid bodies that are slaved in the maximum coordinate direction to the master rigid body. In the part set, see *SET_PART_OPTION, definition the "COLUMN" option may be used to defined as a part attribute the closure distance (D_1 and D_2 in Figure 4.9) which activates the constraint. The constraint does not begin to act until the master rigid body stops. If the distance between the master rigid body is less than or equal to the closure distance, the slave rigid body motion towards the master rigid body also stops. However, the slaved rigid body is free to move away from the master. If the closure distance is input as zero (0.0) then the slaved rigid body stops when the master stops.
PSIDMN	Optional part set ID of rigid bodies that are slaved in the minimum coordinate direction to the master rigid body. In the part set, see *SET_PART_DEFINITION, definition the "COLUMN" option may be used to defined as a part attribute the closure distance (D_1 and D_2 in Figure 4.9) which activates the constraint. The constraint does not begin to act until the master rigid body stops. If the distance between the master rigid body is less than or equal to the closure distance, the slave rigid body motion towards the master rigid body also stops. However, the slaved rigid body is free to move away from the master. If the closure distance is input as zero (0.0) then the slaved rigid body stops when the master stops.
LCVMNX	Load curve ID which defines the maximum absolute value of the velocity as a function of time that is allowed within the stopper. See *DEFINE_ CURVE: EQ.0: no limitation on the velocity.
DIR	Direction stopper acts in: EQ.1: x-translation, EQ.2: y-translation, EQ.3: z-translation, EQ.4: arbitrary, defined by vector VID (see below), EQ.5: x-axis rotation , EQ.6: y-axis rotation, EQ.7: z-axis rotation, EQ.8: arbitrary, defined by vector VID (see below).
VID	Vector for arbitrary orientation of stopper, see *DEFINE_VECTOR.
TB	Time at which stopper is activated.
TD	Time at which stopper is deactivated.

The optional definition of part sets in minimum or maximum coordinate direction allows the motion to be controlled in arbitrary direction.

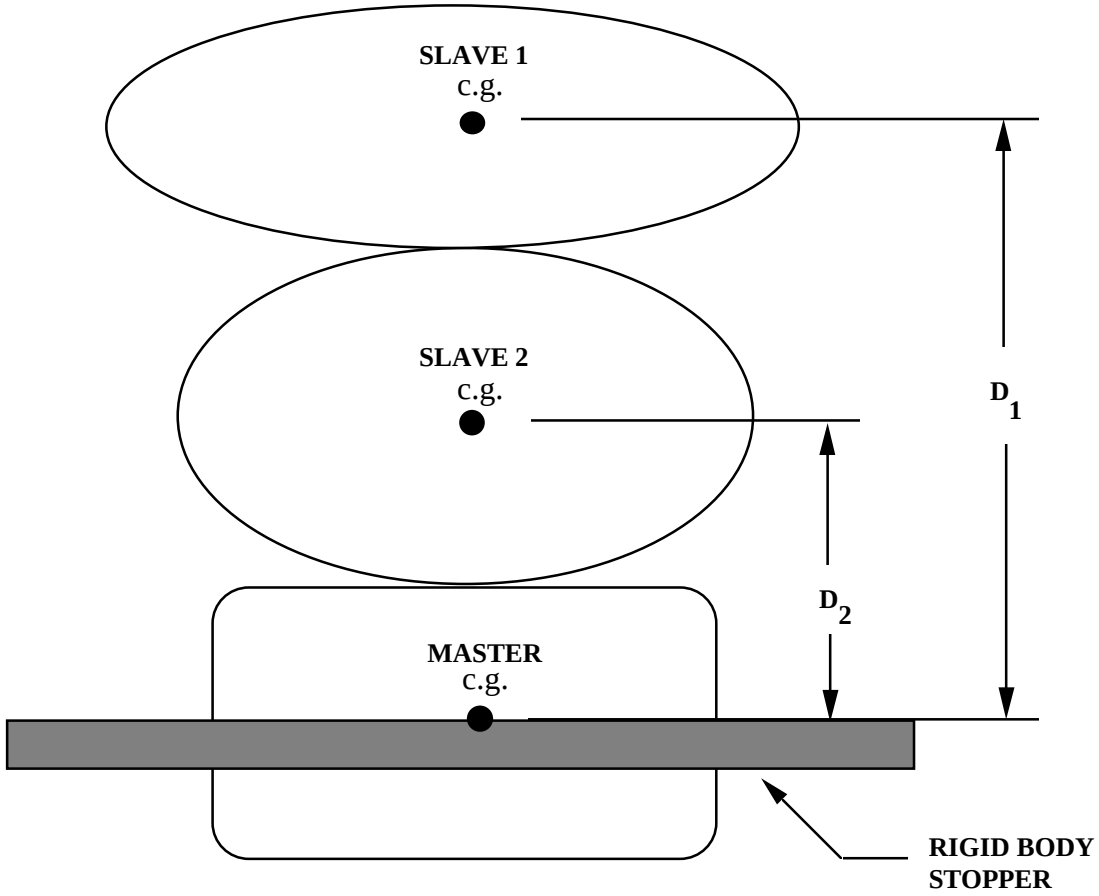


Figure 4.9. When the master rigid body reaches the rigid body stopper, the velocity component into the stopper is set to zero. Slave rigid bodies 1 and 2 also stop if the distance between their mass centers and the master rigid body is less than or equal to the input values D_1 and D_2, respectively. (c.g. ≡ center of gravity).

***CONSTRAINED_RIVET**

Purpose: Define massless rivets between non-contiguous nodal pairs. The nodes must not have the same coordinates. The action is such that the distance between the two nodes is kept constant throughout any motion. No failure can be specified.

Card Format

	1	2	3	4	5	6	7	8
Variable	N1	N2						
Type	I	I						
Default	none	none						
Remarks	1							

VARIABLE	DESCRIPTION
N1	Node ID
N2	Node ID

Remark:

1. Nodes connected by a rivet cannot be members of another constraint set that constrain the same degrees-of-freedom, a tied interface, or a rigid body, i.e., nodes cannot be subjected to multiple, independent, and possibly conflicting constraints. Also care must be taken to ensure that single point constraints applied to nodes in a constraint set do not conflict with the constraint sets constrained degrees-of-freedom.

*CONSTRAINED_SHELL_IN_SOLID

Purpose: Couple a Lagrangian mesh of shells to the material points of an Eulerian flow. This option may also be used to model rebar in concrete or tire cords in rubber. The slave part or slave part set is coupled to the master part or master part set.

Card Format

	1	2	3	4	5	6	7	8
Variable	PSIDS	PSIDM	SSTYP	MSTYP				
Type	I	I	I	I				
Default	none	none	0	0				

VARIABLE	DESCRIPTION
PSIDS	Part or part set ID of embedded shells, see *PART or *SET_PART.
PSIDM	Part or part set ID of solid elements, see *PART or *SET_PART.
SSTYP	Slave type: EQ.0: part set ID, EQ.1: part ID.
MSTYP	Master type: EQ.0: part set ID, EQ.1: part ID.

***CONSTRAINED_SHELL_TO_SOLID**

Purpose: Define a tie between a shell edge and solid elements. Nodal rigid bodies can perform the same function and may also be used.

Card Format

	1	2	3	4	5	6	7	8
Variable	NID	NSID						
Type	I	I						
Default	none	none						
Remarks								

VARIABLE	DESCRIPTION
NID	Shell node ID
NSID	Solid nodal set ID, see *SET_NODE_OPTION.

The shell-brick interface, an extension of the tied surface capability, ties regions of hexahedron elements to regions of shell elements. A shell node may be tied to up to nine brick nodes lying along the tangent vector to the nodal fiber. See Figure 4.10. During the calculation, nodes thus constrained must lie along the fiber but can move relative to each other in the fiber direction. The brick nodes must be input in the order in which they occur, in either the plus or minus direction, as one moves along the shell node fiber.

This feature is intended to tie four node shells to eight node shells or solids; it is not intended for tying eight node shells to eight node solids.

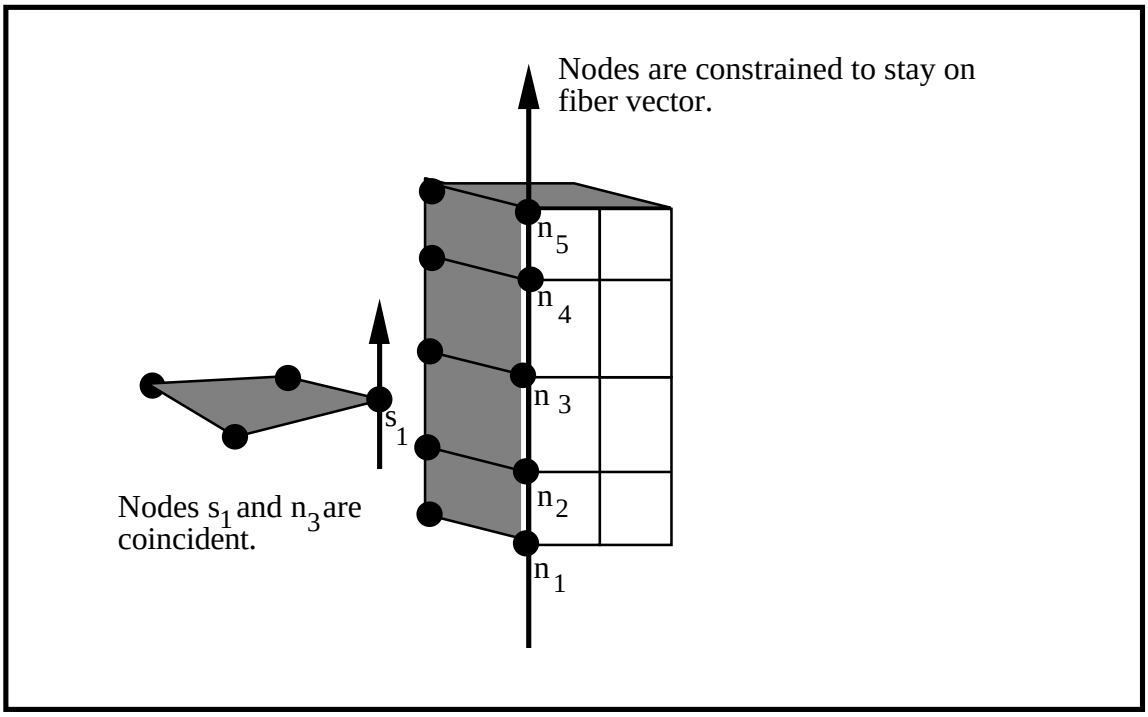


Figure 4.10. The interface between shell elements and solids ties shell node s1 to a line of nodes on the solid elements n1-n5. It is very important for the nodes to be aligned.

***CONSTRAINED_SPOTWELD**

Purpose: Define massless spotwelds between non-contiguous nodal pairs. The nodes must not have the same coordinates. A rigid beam is assumed between the nodal pairs, thus nodal rotations and displacements are coupled. A brittle failure can be specified.

Card Format

	1	2	3	4	5	6	7	8
Variable	N1	N2	SN	SS	N	M		
Type	I	I	F	F	F	F		
Default	none	none	optional	optional	optional	optional		
Remarks	1.		2.					

VARIABLE	DESCRIPTION
N1	Node ID
N2	Node ID
SN	Normal force at spotweld failure (see Remark 2 below).
SS	Shear force at spotweld failure (see Remark 2 below).
N	Exponent for normal spotweld force (see Remark 2 below).
M	Exponent for shear spotweld force (see Remark 2 below).

Remarks:

1. Nodes connected by a spotweld cannot be members of another constraint set that constrain the same degrees-of-freedom, a tied interface, or a rigid body, i.e., nodes cannot be subjected to multiple, independent, and possibly conflicting constraints. Also, care must be taken to ensure that single point constraints applied to nodes in a constraint set do not conflict with the constraint sets constrained degrees-of-freedom.

2. Failure of the spotwelds occurs when:

$$\left(\frac{|f_n|}{S_n}\right)^n + \left(\frac{|f_s|}{S_s}\right)^m \geq 1$$

where f_n and f_s are the normal and shear interface force. Component f_n is nonzero for tensile values only.

***CONSTRAINED_TIE-BREAK**

Purpose: Define a tied shell edge to shell edge interface that can release locally as a function of plastic strain of the shells surrounding the interface nodes. A rather ductile failure is achieved.

Card Format

	1	2	3	4	5	6	7	8
Variable	SNSID	MNSID	EPPF					
Type	I	I	F					
Default	none	none	0.					
Remarks								

VARIABLE	DESCRIPTION
SNSID	Slave node set ID, see *SET_NODE_OPTION.
MNSID	Master node set ID, see *SET_NODE_OPTION.
EPPF	Plastic strain at failure

Remarks:

1. The plastic strain taken for the failure criteria is computed as an average volume weighted plastic strain from the shell elements surrounding both node sets. Each node set is considered separately when the plastic strains are computed.

*CONSTRAINED_TIED_NODES_FAILURE

Purpose: Define a tied node set with failure based on plastic strain. The nodes must be coincident.

Card Format

	1	2	3	4	5	6	7	8
Variable	NSID	EPPF						
Type	I	F						
Default	none	0.						
Remarks								

VARIABLE	DESCRIPTION
NSID	Nodal set ID, see *SET_NODE_OPTION.
EPPF	Plastic strain at failure

This feature applies only to thin shell elements. The specified nodes are tied together until the average volume weighted plastic strain exceeds the specified value. Entire regions of individual shell elements may be tied together unlike the tie-breaking shell slidelines. The tied nodes are coincident until failure.

*CONTACT

***CONTACT_{OPTION1}_{OPTION2}_{OPTION3}**

Purpose: Define a sliding contact interface.

OPTION1 specifies the contact type (also, see remarks 1-3 below) :

AIRBAG_SINGLE_SURFACE
AUTOMATIC_NODES_TO_SURFACE
AUTOMATIC_ONE_WAY_SURFACE_TO_SURFACE
AUTOMATIC_SINGLE_SURFACE
AUTOMATIC_SURFACE_TO_SURFACE
CONSTRAINT_NODES_TO_SURFACE
CONSTRAINT_SURFACE_TO_SURFACE
DRAWBEAD
ERODING_NODES_TO_SURFACE
ERODING_SINGLE_SURFACE
ERODING_SURFACE_TO_SURFACE
NODES_TO_SURFACE
ONE_WAY_SURFACE_TO_SURFACE
RIGID_NODES_TO_RIGID_BODY
RIGID_BODY_ONE_WAY_TO_RIGID_BODY
RIGID_BODY_TWO_WAY_TO_RIGID_BODY
SINGLE_EDGE
SINGLE_SURFACE
SLIDING_ONLY
SLIDING_ONLY_PENALTY
SURFACE_TO_SURFACE
TIEBREAK_NODES_TO_SURFACE
TIEBREAK_SURFACE_TO_SURFACE
TIED_NODES_TO_SURFACE
TIED_SHELL_EDGE_TO_SURFACE
TIED_SURFACE_TO_SURFACE

OPTION2 specifies a thermal contact and takes the single option:

THERMAL

OPTION3 specifies that the first card to read defines the title and ID number of contact interface and takes the single option:

TITLE

Note: OPTION1, OPTION2, and OPTION3 may appear in any order.

At present, the contact ID number and title are ignored by LS-DYNA3D but are included for extension in the near future. The title card is picked up by some of the peripheral LS-DYNA3D codes to aid in post-processing.

The keyword options for the contact type and the corresponding Version 92X type numbers are:

STRUCTURED INPUT TYPE ID	KEYWORD NAME
a13	AIRBAG_SINGLE_SURFACE
a 5	AUTOMATIC_NODES_TO_SURFACE
a10	AUTOMATIC_ONE_WAY_SURFACE_TO_SURFACE
13	AUTOMATIC_SINGLE_SURFACE
a 3	AUTOMATIC_SURFACE_TO_SURFACE
18	CONSTRAINT_NODES_TO_SURFACE
17	CONSTRAINT_SURFACE_TO_SURFACE
23	DRAWBEAD
16	ERODING_NODES_TO_SURFACE
14	ERODING_SURFACE_TO_SURFACE
15	ERODING_SINGLE_SURFACE
5	NODES_TO_SURFACE
10	ONE_WAY_SURFACE_TO_SURFACE
20	RIGID_NODES_TO_RIGID_BODY
21	RIGID_BODY_ONE_WAY_TO_RIGID_BODY
19	RIGID_BODY_TWO_WAY_TO_RIGID_BODY
22	SINGLE_EDGE
4	SINGLE_SURFACE
1	SLIDING_ONLY
p 1	SLIDING_ONLY_PENALTY
3	SURFACE_TO_SURFACE
8	TIEBREAK_NODES_TO_SURFACE
9	TIEBREAK_SURFACE_TO_SURFACE
6	TIED_NODES_TO_SURFACE
7	TIED_SHELL_EDGE_TO_SURFACE
2	TIED_SURFACE_TO_SURFACE

Remarks:

1. TIED_NODES_TO_SURFACE
   TIED_SHELL_EDGE_TO_SURFACE
   TIED_SURFACE_TO_SURFACE

   These contact definitions are based on constraint equations and will not work with rigid bodies. It is suggested to use the CONSTRAINED_EXTRA_NODE_OPTION instead.

2. CONSTRAINT_NODES_TO_SURFACE
   CONSTRAINT_SURFACE_TO_SURFACE

   These contact definitions have to be used with care. The surface and the nodes which are constrained to a surface are not allowed to be used in any other CONSTRAINT_... contact definition. If, however, contact has to be defined from both sides as in sheetmetalforming, one of these contact definitions can be a CONSTRAINT_ type; the other one could be a standard penalty type such as SURFACE_TO_SURFACE or NODES_TO_SURFACE.

3. AIRBAG_SINGLE_SURFACE
   AUTOMATIC_NODES_TO_SURFACE
   AUTOMATIC_ONE_WAY_SURFACE_TO_SURFACE
   AUTOMATIC_SINGLE_SURFACE
   AUTOMATIC_SURFACE_TO_SURFACE
   SINGLE_SURFACE

   These contact definitions require thickness to be taken into account for rigid bodies modeled with shell elements. Therefore, care should be taken to ensure that realistic thicknesses are specified for the rigid body shells. A thickness that is too small may result in loss of contact and an unrealistically large thickness may result in a degradation in speed during the bucket sorts as well as nonphysical behavior. The SHLTHK option on the *CONTROL_CONTACT card is ignored for these contact types.

4. Two methods are used in LS-DYNA3D for projecting the contact surface to account for shell thicknesses. The choice of methods can influence the accuracy and cost of the calculation. Segment based projection is used in contact types:

   AIRBAG_SINGLE_SURFACE
   AUTOMATIC_NODES_TO_SURFACE
   AUTOMATIC_ONE_WAY_SURFACE_TO_SURFACE
   AUTOMATIC_SINGLE_SURFACE
   AUTOMATIC_SURFACE_TO_SURFACE

The remaining contact types use nodal normal projections if projections are used. The main advantage of nodal projections is that a continuous contact surface is obtained which is much more accurate in applications such as metal forming. The disadvantages of nodal projections are the higher costs due to the nodal normal calculations, difficulties in treating T-intersections and other geometric complications, and the need for consistent orientation of contact surface segments. The contact type:

SINGLE_SURFACE

uses nodal normal projections and consequently is slower than the alternatives.

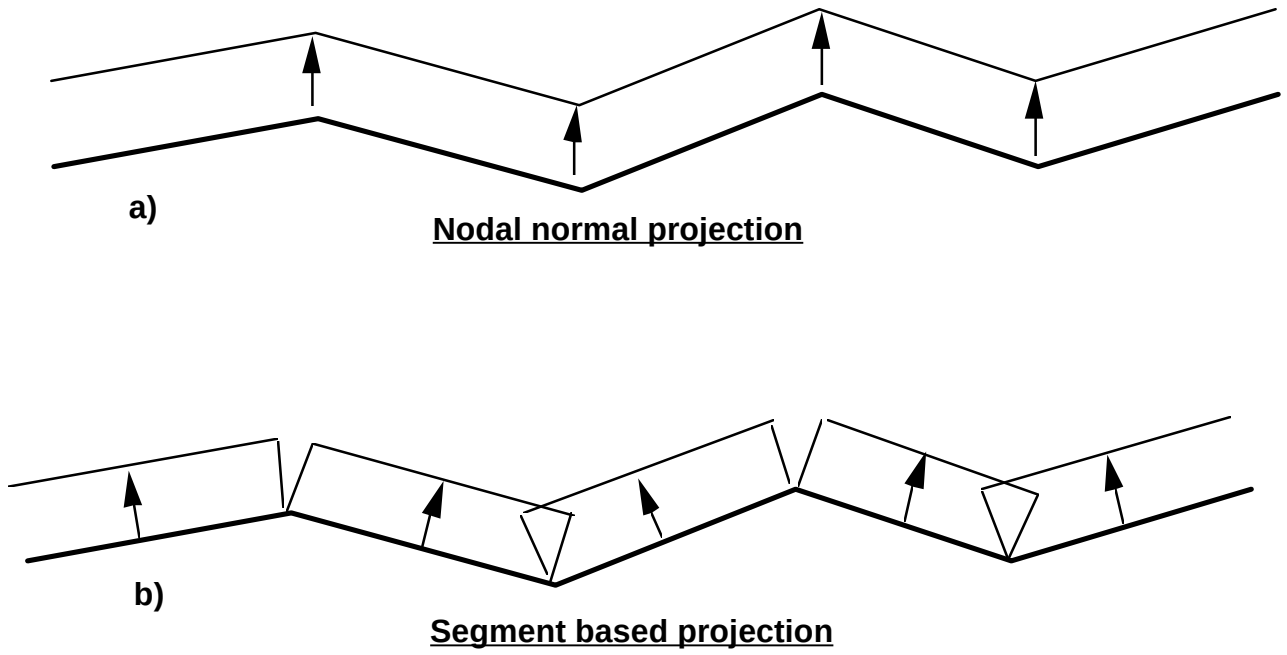


Figure 5.1. Nodal normal and segment based projection is used in the contact options.

*CONTACT

Read the following card here if and only if the option **TITLE** is specified:

Optional 1 2

Variable	CID	NAME
Type	I	A70

Card Format

Cards 1 to 3 are mandatory for all contact types.

Card 1 1 2 3 4 5 6 7 8

Variable	SSID	MSID	SSTYP	MSTYP	SBOXID	MBOXID	SPR	MPR
Type	I	I	I	I	I	I	I	I
Default	none	none	none	none			0	0
Remarks	1	2			optional	optional	0=off	0=off

Card 2 1 2 3 4 5 6 7 8

Variable	FS	FD	DC	VC	VDC	PENCHK	BT	DT
Type	F	F	F	F	F	I	F	F
Default	0.	0.	0.	0.	0.	0	0.	1.0E20
Remarks								

The variables FSF and VSF below can be overridden segment by segment on the *SET_SEGMENT or *SET_SHELL_ OPTION cards for the **slave surface only** as A3 and A4, and for the **master surface only** as A1 and A2. See *SET_SEGMENT and *SET_SHELL_OPTION.

Card 3	1	2	3	4	5	6	7	8
Variable	SFS	SFM	SST	MST	SFST	SFMT	FSF	VSF
Type	F	F	F	F	F	F	F	F
Default	1.	1.	element thickness	element thickness	1.	1.	1.	1.
Remarks								

Additional Card required for CONSTRAINT_contact.

Card 4	1	2	3	4	5	6	7	8
Variable	KPF							
Type	F							
Default	0.0							

Additional Card required for DRAWBEAD contact.

Card 4	1	2	3	4	5	6	7	8
Variable	LCIDRF	LCIDNF	DBDTH	DFSCL	NUMINT			
Type	I	I	F	F	I			
Default	required	required	0.0	1.0	0			

Additional Card required for ERODING_contact.

Card 4	1	2	3	4	5	6	7	8
Variable	ISYM	EROSOP	IADJ					
Type	I	I	I					
Default	0	1	0					

Additional Card required for TIEBREAK_NODE_contact.

These attributes can be overridden node by node on the *SET_NODE_OPTION cards.

Card 4	1	2	3	4	5	6	7	8
Variable	NFLF	SFLF	NEN	MES				
Type	F	F	F	F				
Default	required	required	2.	2.				

Additional Card required for TIEBREAK_SURFACE_contact.

These attributes can be overridden segment by segment on the *SET_SEGMENT or *SET_SHELL_ OPTION cards for the **slave surface only** as A1 and A2. These variables do not apply to the master surface.

Card 5	1	2	3	4	5	6	7	8
Variable	NFLS	SFLS	TBLCID					
Type	F	F	I					
Default	required	required	0					

Additional Card required for RIGID_contact.

Card 5	1	2	3	4	5	6	7	8
Variable	LCID	FCM	US					
Type	I	I	F					
Default	required	required	from LCID					

The following card is ready if the THERMAL option is specified.

Variable	CF	RAD	HTC	GCRIT	GMAX			
Type	F	F	F	F	F			

These two optional cards are read unless an "*" card is found. Either the first or the first and second optional cards may be defined. The second card may not be defined independently.

Optional Card	1	2	3	4	5	6	7	8
Variable	SOFT	SOFSCL	LCIDAB	MAXPAR	PENTOL	DEPTH	BSORT	FRCFRQ
Type	I	F	I	F	F	I	I	I
Default	0	.1	0	1.025.	0.	2	10-100	1
Remarks			type a13			type 13		

*CONTACT

Optional Card	1	2	3	4	5	6	7	8
Variable	PENMAX	THKOPT	SHLTHK					
Type	F	I	I					
Default	0	0	0					
Remarks	3, 5, 10	3, 5, 10	3, 5, 10					

VARIABLE	DESCRIPTION
CID	Contact interface ID. This must be a unique number.
NAME	Interface descriptor. It is suggested that unique descriptions are used.
SSID	Slave segment, node set ID, partset ID, part ID, or shell element set ID, see *SET_SEGMENT, *SET_NODE_OPTION, *PART, *SET_PART or *SET_SHELL_OPTION: EQ.0: all segments are included for single surface contact.
MSID	Master segment set ID, partset ID, part ID, or shell element set ID, see *SET_SEGMENT, *SET_NODE_OPTION, *PART, *SET_PART, or *SET_SHELL_OPTION: EQ.0: for single surface contact.
SSTYP	Slave segment or node set type. The type must correlate with the number specified for SSID: EQ.0: segment set ID for surface to surface contact, EQ.1: shell element set ID for surface to surface contact, EQ.2: part set ID, EQ.3: part ID, EQ.4: node set ID for node to surface contact, EQ.5: include all for single surface defintion. EQ.6: part set ID for exempted parts. All non exempted parts are included in the contact.
MSTYP	Master segment set type. The type must correlate with the number specified for MSID: EQ.0: segment set ID, EQ.1: shell element set ID, EQ.2: part set ID, EQ.3: part ID.

VARIABLE	DESCRIPTION
SBOXID	BOXID, Include only slave nodes/segments within specified box, see *DEFINE_BOX, in contact definition. Nodes/shell/elements/segments/ parts as defined by SSID are taken.
MBOXID	BOXID, Include only master segments within specified box, see *DEFINE_BOX, in contact. Shell elements/segments/parts as defined by MSID are taken.
SPR	Include slave side in printed and binary force interface file: EQ.1: slave side forces included.
MPR	Include master side in printed and binary force interface file: EQ.1: master side forces included.
FS	Static coefficient of friction. The functional coefficient is assumed to be dependent on the relative velocity v_{rel} of the surfaces in contact $\mu_c = FD + (FS - FD)e^{-DC \cdot 1v^1_{rel}}$.
FD	Dynamic coefficient of friction. The functional coefficient is assumed to be dependent on the relative velocity v_{rel} of the surfaces in contact $\mu_c = FD + (FS - FD)e^{-DC \cdot 1v^1_{rel}}$.
DC	Exponential decay coefficient. The functional coefficient is assumed to be dependent on the relative velocity v_{rel} of the surfaces in contact $\mu_c = FD + (FS - FD)e^{-DC \cdot 1v^1_{rel}}$.
VC	Coefficient for viscous friction. This is necessary to limit the friction force to a maximum. A limiting force is computed $F_{lim} = VC \cdot A_{cont}$. A_{cont} being the area of the segment contacted by the node in contact. The suggested value for VC is to use the yield stress in shear $VC = \frac{\sigma_o}{\sqrt{3}}$ where σ_o is the yield stress of the contacted material.
VDC	Viscous damping coefficient in percent of critical. In order to avoid undesirable oscillation in contact, e.g., for sheet forming simulation, a contact damping perpendicular to the contacting surfaces is applied. Damping coefficient $\xi = \frac{VDC}{1DO} \cdot \xi_{crit} \cdot \varepsilon g \cdot VDC = 20$. ξ_{crit} is determined in the following fashion by LS-DYNA3D. $\xi_{crit} = 2mw; \quad m = min(m_{slave}, m_{master})$ *mass of master resp. slave node* $w = \sqrt{k \cdot \frac{m_{slave} + m_{master}}{m_{slave} \cdot m_{master}}}$ k interface stiffness

VARIABLE	DESCRIPTION
PENCHK	Small penetration in contact search option. If the slave node penetrates more than the segment thickness times the factor XPENE, see *CONTROL_ CONTACT, the penetration is ignored and the slave node is set free. The thickness is taken as the shell thickness if the segment belongs to a shell element or it is taken as 1/20 of its shortest diagonal if the segment belongs to a solid element. This option applies to the surface to surface contact algorithms: EQ.0: check is turned off, EQ.1: check is turned on, EQ.2: check is on but shortest diagonal is used.
BT	Birth time (contact surface becomes active at this time).
DT	Death time (contact surface is deactivated at this time).
SFS	Scale factor on default slave penalty stiffness, see also *CONTROL_ CONTACT.
SFM	Scale factor on default master penalty stiffness, see also *CONTROL_ CONTACT.
SST	Optional thickness for slave surface (overrides true thickness). This option applies only to contact with shell elements. True thickness is the element thickness of the shell elements.
MST	Optional thickness for master surface (overrides true thickness). This option applies only to contact with shell elements. True thickness is the element thickness of the shell elements.
SFST	Scale factor for slave surface thickness (scales true thickness). This option applies only to contact with shell elements. True thickness is the element thickness of the shell elements.
SFMT	Scale factor for master surface thickness (scales true thickness). This option applies only to contact with shell elements. True thickness is the element thickness of the shell elements.
FSF	Coulomb friction scale factor. The Coulomb friction value is scaled as $\mu_{sc} = FSF \cdot \mu_c$, see above.
VSF	Viscous friction scale factor. If this factor is defined then the limiting force becomes: $F_{lim} = VSF \cdot VC \cdot A_{cont}$, see above.
KPF	Kinematic partition factor for constraint: EQ. 0.0: fully symmetric treatment. EQ.1.0: one way treatment with slave nodes constrained to master surface. Only the slave nodes are checked against contact. EQ.-1.0: one way treatment with master nodes constrained to slave surface. Only the master nodes are checked against contact.

VARIABLE	DESCRIPTION
LCIDRF	Load curve ID giving the bending component of the restraining force, $F_{bending}$, per unit draw bead length as a function of displacement, δ, see Figure 5.2. This force is due to the bending and unbending of the blank as it moves through the drawbead. The total restraining force is the sum of the bending and friction components.
LCIDNF	Load curve ID giving the normal force per unit draw bead length as a function of displacement, δ, see Figure 5.2. This force is due to the bending of the blank into the draw bead as the binder closes on the die and represents a limiting value. The normal force begins to develop when the distance between the die and binder is less than the draw bead depth. *As the binder and die close on the blank this force should diminish or reach a plateau,* see the explanation below.
DBDTH	Draw bead depth, see Figure 5.2. Necessary to determine correct δ displacement from contact displacements.
DFSCL	Scale factor for load curve. Default=1.0. This factor scales load curve ID, LCIDRF above.
NUMINT	Number of equally spaced integration points along the draw bead: EQ.0: Internally calculated based on element size of elements that interact with draw bead. This is necessary for the correct calculation of the restraining forces. More integration points may increase the accuracy since the force is applied more evenly along the bead.
ISYM	Symmetry plane option: EQ.0: off, EQ.1: do not include faces with normal boundary constraints (e.g., segments of brick elements on a symmetry plane). This option is important to retain the correct boundary conditions in the model with symmetry.
EROSOP	Erosion/Interior node option: EQ.0: only exterior boundary information is saved, EQ.1: storage is allocated so that eroding contact can occur. Otherwise, no contact is assumed after erosion of the corresponding element.
IADJ	Adjacent material treatment for solid elements: EQ.0: solid element faces are included only for free boundaries, EQ.1: solid element faces are included if they are on the boundary of the material subset. This option also allows the erosion within a body and the consequent treatment of contact.
NFLF	Normal failure force. Only tensile failure, i.e., tensile normal forces, will be considered in the failure criterion.
SFLF	Shear failure force

VARIABLE	DESCRIPTION
NEN	Exponent for normal force
MES	Exponent for shear force. Failure criterion: $\left(\frac{\lvert f_n \rvert}{NFLF}\right)^{NEN} + \left(\frac{\lvert f_s \rvert}{SFLF}\right)^{MES} \geq 1.$ Failure is assumed if the left side is larger than 1. 4*n* and 4*s* are the normal and shear interface force.
NFLS	Normal failure stress
SFLS	Shear failure stress. Failure criterion: $\left(\frac{\lvert \sigma_n \rvert}{NFLS}\right)^2 + \left(\frac{\lvert \sigma_s \rvert}{SFLS}\right)^2 \geq 1.$
TBLCID	Optional load curve number defining the resisting stress versus gap opening for the post failure response. This can be used to model the failure of adhesives.
LCID	Load curve ID giving force versus deflection behavior for RIGID_ contact. See also the definition of FCM below.
FCM	Force calculation method for RIGID_contact: EQ.1: Load curve gives total normal force on surface versus maximum penetration of any node (RIGID_BODY_ONE_WAY only). EQ.2: Load curve gives normal force on each node versus penetration of node through the surface (all RIGID_contact types). EQ.3: Load curve gives normal pressure versus penetration of node through the surface (RIGID_BODY_TWO_WAY and RIGID_BODY_ONE_WAY only).
US	Unloading stiffness for RIGID_contact. The default is to unload along the loading curve. This should not be larger than the maximum value used in the loading curve.
CF	Conductivity of fluid in gap
RAD	Radiation conductance across gap
HTC	Heat transfer coefficient
GCRIT	Critical gap. For gaps less than GCRIT the heat transfer coefficient is used. For gaps greater than GCRIT the conductivity of fluid in the gap is used.
GMAX	No thermal contact if gap is greater than this value.

VARIABLE	DESCRIPTION
SOFT	Soft constraint option: EQ.0: penalty formulation, EQ.1: soft constraint formulation. Necessary if a surface in contact has wildly varying stiffnesses along surface.
SOFSCL	Scale factor for constraint forces of soft constraint option (default=.10). Values greater than .5 for single surface contact and 1.0 for a one way treatment are inadmissible.
LCIDAB	Load curve ID defining airbag thickness as a function of time for type a13 contact (*CONTACT_AIRBAG_SINGLE_SURFACE).
MAXPAR	Maximum parametric coordinate in segment search (values 1.025 and 1.20 recommended). Larger values can increase cost. If zero, the default is set to 1.025. This factor allows an increase in the size of the segments. May be useful at sharp corners.
PENTOL	Special penetration tolerance (not currently used).
DEPTH	Search depth in automatic contact. Value of 1 is sufficiently accurate for most crash applications and is much less expensive. LS-DYNA3D for improved accuracy sets this value to 2. If zero, the default is set to 2.
BSORT	Number of cycles between bucket sorts. Values of 25 and 100 are recommended for contact types 4 and 13 (SINGLE_SURFACE), respectively. Values of 10-15 are okay for the surface to surface and node to surface contact. If zero, LS-DYNA3D determines the interval.
FRCFRQ	Number of cycles between contact force updates for penalty.contact formulations. This option can provide a significant speed-up of the contact treatment. If used, values exceeding 3 or 4 are dangerous. Considerable care must be exercised when using this option, as this option assumes that contact does not change FRCFRG cycles. EQ.0: FRCFRG is set to 1 and force calculations are performed each cycle-strongly recommended.
PENMAX	Maximum penetration distance for old type 3, 5, and 10 contact, see page 5.3 (Also, see Table 5.1): EQ.0: Use small penetration search and value calculated from thickness and XPENE, see *CONTROL_CONTACT. GT.0: Ignore element thickness and XPENE, use this value instead. Note PENCHK must be set to zero, see above on Card 2.
THKOPT	Thickness option for contact types 3, 5, and 10: EQ.0: default is taken from control card, *CONTROL_CONTACT, EQ.1: thickness offsets are included, EQ.2: thickness offsets are not included (old way).

VARIABLE	DESCRIPTION
SHLTHK	Define if and only if THKOPT above equals 1. Shell thickness considered in type surface to surface and node to surface type contact options, where options 1 and 2 below activate the new contact algorithms. The thickness offsets are always included in single surface and constraint method contact types: EQ.0: thickness is not considered, EQ.1: thickness is considered but rigid bodies are excluded, EQ.2: thickness is considered including rigid bodies.
SNLOG	Disable shooting node logic in thickness offset contact: EQ.0: logic is enabled (default), EQ.1: logic is skipped (sometimes recommended for metal forming calculations).

Remarks:

1. Giving a slave set ID equal to zero is valid only for the single surface contact algorithms, i.e., the options SINGLE_SURFACE, and the AUTOMATIC_, AIRBAG_, and ERODING_ SINGLE_ SURFACE options.

2. A master set ID is not defined for the single surface contact algorithms.

The draw bead is defined by a *consecutive* list of nodes that lie along the draw bead. For straight draw beads only two nodes need to be defined, i.e., one at each end, but for curved beads sufficient nodes are required to define the curvature of the bead geometry. The integration points along the bead are equally spaced and are independent of the nodal spacing used in the definition of the draw bead. By using the capability of tying extra nodes to rigid bodies (see *CONSTRAINED_EXTRA_NODES_OPTION) the draw bead nodal points do not need to belong to the element connectivities of the die and binder.

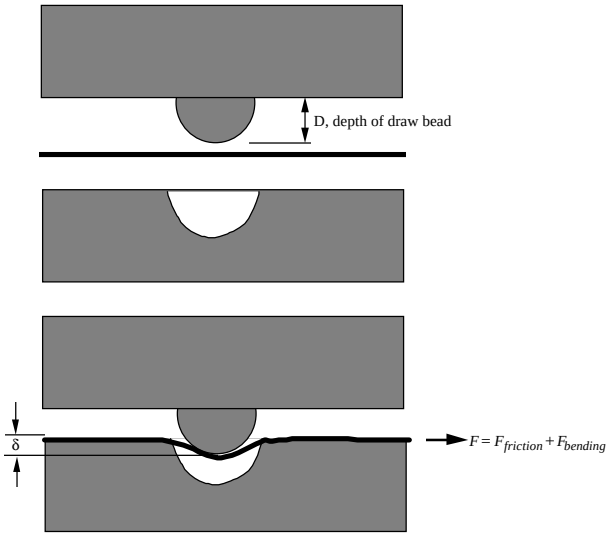


Figure 5.2. Draw bead contact model defines a resisting force as a function of draw bead displacement. The friction force is computed from the normal force in the draw bead and the given friction coefficient.

INTERFACE TYPE ID	PENCHK	ELEMENT TYPE	FORMULA FOR RELEASE OF PENETRATING NODAL POINT
1, 2, 6, 7	———	———	——————----------
3, 5, 8, 9, 10	0	solid	d=PENMAX if and only if PENMAX>0
(without thickness)			d=1.e+10 if PENMAX=0
		shell	d=PENMAX if and only if PENMAX>0
			d=1.e+10 if PENMAX=0
	1	solid	d=XPENE*thickness of solid element
		shell	d=XPENE*thickness of shell element
	2	solid	d=0.05*minimum diagonal length
		shell	d=0.05*minimum diagonal length
3, 5, 10 (thickness)	———	solid	d=XPENE*thickness of solid element
17, and 18		shell	d=XPENE*thickness of shell element
a3, a5, a10	———	solid	d=0.5*thickness of solid element
		shell	d=0.4*(slave thickness+master thickness)
4, 13, 15	———	solid	d=0.5*thickness of solid element
		shell	d=0.4*(slave thickness+master thickness)

Table 5.1. Criterion for node release for nodal points which have penetrated too far. Larger penalty stiffnesses are recommended for the contact interface which allows nodes to be released. For node-to-surface type contacts (5, 5a) the element thicknesses which contain the node determines the nodal thickness. The parameter is defined on the *CONTROL_CONTACT input.

***CONTACT_ENTITY**

Purpose: Define a contact entity. Geometric contact entities treat the impact between a deformable body defined as a set of slave nodes or nodes in a shell part set and a rigid body. The shape of the rigid body is determined by attaching geometric entities. Contact is treated between these geometric entities and the slave nodes using a penalty formulation. The penalty stiffness is optionally maximized within the constraint of the Courant criterion. As an alternative, a finite element mesh made with shells can be used as geometric entity. Also, axisymmetric entities with arbitrary shape made with multilinear polygons are possible. The latter is particularly useful for metalforming simulations.

Card Format

Card 1

Variable	1	2	3	4	5	6	7	8
	PID	GEOTYP	SSID	SSTYP	SF	DF	CF	INTORD
Type	I	I	I	I	F	F	F	I
Default	required	required	required	0	1.	0.	0.	0

Card 2

Variable	1	2	3	4	5	6	7	8
	BT	DT	SO	GO				
Type	F	F	I	I				
Default	0.	1.E+20	0	0				

Card 3	1	2	3	4	5	6	7	8
Variable	XC	YC	ZC	AX	AY	AZ		
Type	F	F	F	F	F	F		
Default	0.	0.	0.	0.	0.	0		

Card 4	1	2	3	4	5	6	7	8
Variable	BX	BY	BZ					
Type	F	F	F					
Default	0.	0.	0.					

Card 5	1	2	3	4	5	6	7	8
Variable	INOUT	G1	G2	G3	G4	G5	G6	G7
Type	I	F	F	F	F	F	F	F
Default	0	0.	0.	0.	0.	0.	0.	0.

VARIABLE	DESCRIPTION
PID	Part ID of the rigid body to which the geometric entity is attached, see *PART.
GEOTYP	Type of geometric entity: EQ.1: plane, EQ.2: sphere, EQ.3: cylinder, EQ.4: ellipsoid, EQ.5: torus, EQ.6: CAL3D/MADYMO Plane, see Appendix F, EQ.7: CAL3D/MADYMO Ellipsoid, see Appendix F, EQ.8: VDA surface, see Appendix J, EQ.9: rigid body finite element mesh (shells only), EQ.10: finite plane, EQ.11: load curve defining line as surface profile of axisymmetric rigid bodies.
SSID	Slave set ID, see *SET_NODE_OPTION, *PART, or *SET_PART.
SSTYP	Slave set type: EQ.0: node set, EQ.1: part ID, EQ.2: part set ID.
SF	Penalty scale factor. Useful to scale maximized penalty.
DF	Damping option, see description for *CONTACT_OPTION: EQ..0: no damping, GT..0: viscous damping in percent of critical, e.g., 20 for 20% damping, EQ.-n: |n| is the load curve ID giving the damping force versus relative normal velocity (see comment below).
CF	Coulomb friction coefficient. Assumed to be constant.
INTORD	Integration order (slaved materials only). This option is not available with entity types 8 and 9 where only nodes are checked: EQ.0: check nodes only, EQ.1: 1 point integration over segments, EQ.2: 2×2 integration, EQ.3: 3×3 integration, EQ.4: 4×4 integration, EQ.5: 5×5 integration. This option allows a check of the penetration of the rigid body into the deformable (slaved) material. Then virtual nodes at the location of the integration points are checked.
BT	Birth time
DT	Death time

VARIABLE	DESCRIPTION
SO	Flag to use penalty stiffness as in surface to surface contact: EQ.0: contact entity stiffness formulation, EQ.1: surface to surface contact method, EQ.-n: |n| is the load curve ID giving the force versus the normal penetration.
GO	Flag for mesh generation of the contact entity for entity types 1-5 and 10-11. This is used for visualization in post-processing only. EQ.0: mesh is not generated, EQ.1: mesh is generated.
XC	x-center, x_c, see comments below.
YC	y-center, y_c, see comments below.
ZC	z-center, z_c. See comments below.
AX	x-direction for local axis A, A_x, see comments below.
AY	y-direction for local axis A, A_y, see comments below.
AZ	z-direction for local axis A, A_z, see comments below.
BX	x-direction for local axis B, B_x, see comments below.
BY	y-direction for local axis B, B_y,. see comments below.
BZ	z-direction for local axis B, B_z,. see comments below.
INOUT	In-out flag. Allows contact from the inside or the outside (default) of the entity: EQ.0: slave nodes exist outside of the entity, EQ.1: slave nodes exist inside the entity.
G1	Entity coefficient g_1 (CAL3D/MADYMO plane or ellipse number) for coupled analysis (see Appendix F).
G2	Entity coefficient g_2, see comments below.
G3	Entity coefficient g_3, see comments below.
G4	Entity coefficient g_4, see comments below.
G5	Entity coefficient g_5, see comments below.
G6	Entity coefficient g_6, see comments below.
G7	Entity coefficient g_7, see comments below.

The optional load curves that are defined for damping versus relative normal velocity and for force versus normal penetration should be defined in the positive quadrant. The sign for the damping force depends on the direction of the relative velocity and the treatment is symmetric if the damping curve is in the positive quadrant. If the damping force is defined in the negative and positive quadrants, the sign of the relative velocity is used in the table look-up.

The coordinates, (x_c, y_c, z_c), are the positions of the local origin of the geometric entity in global coordinates. The entity's local A-axis is determined by the vector (A_x, A_y, A_z) and the local B-axis by the vector (B_x, B_y, B_z).

Cards 3 and 4 define a local to global transformation. The geometric contact entities are defined in a local system and transformed into the global system. For the ellipsoid, this is necessary because it has a restricted definition for the local position. For the plane, sphere, and cylinder, the entities can be defined in the global system and the transformation becomes (x_c, y_c, z_c)=(0,0,0), (A_x, A_y, A_z)=(1,0,0), and (B_x, B_y, B_z)=(0,1,0).

Figures 5.3a and 5.3b show the definitions of the geometric contact entities. The relationships between the entity coefficients and the Figure 5.3a and 5.3b variables are as follows (please note that (P_x,P_y,P_z) is a position vector and that (Q_x,Q_y,Q_z) is a direction vector):

GEOTYP = 1:	g1 = Px	g4 = Qx
	g2 = Py	g5 = Qy
	g3 = Pz	g6 = Qz
		g7 = L

If automatic generation is used, a square plane of length L on each edge is generated which represents the infinite plane. If generation is inactive, then g7 may be ignored.

GEOTYP = 2:	g1 = Px	g4 = r
	g2 = Py	
	g3 = Pz	

GEOTYP = 3:	g1 = Px	g4 = Qx
	g2 = Py	g5 = Qy
	g3 = Pz	g6 = Qz
		g7 = r

If automatic generation is used, a cylinder of length $\sqrt{Qx^2 + Qy^2 + Qz^2}$ and radius r is generated which represents the infinite cylinder.

GEOTYP = 4: g1 = Px
g2 = Py
g3 = Pz
g4 = a
g5 = b
g6 = c
g7 = *n* (order of the ellipsoid)

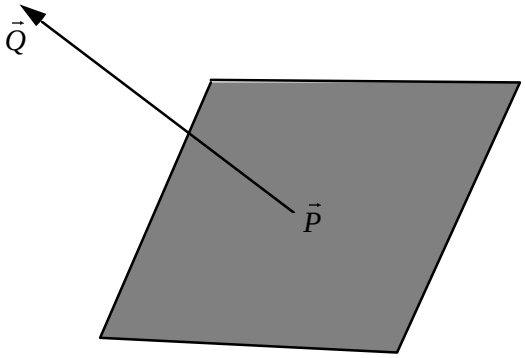
GEOTYP = 5: g1 = Radius of torus
g2 = r

GEOTYP = 8: g1 = Blank thickness (option to override true thickness)
g2 = Scale factor for true thickness (optional)
g3 = Load curve ID defining thickness versus time. (optional)

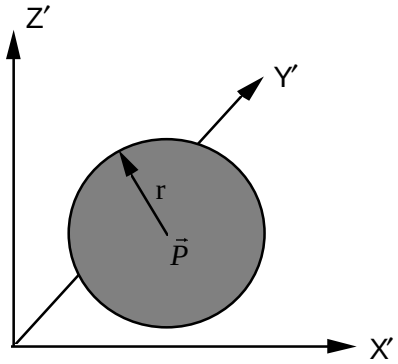
GEOTYP = 9: g1 = Shell thickness (option to override true thickness)
g2 = Scale factor for true thickness (optional)
g3 = Load curve ID defining thickness versus time. (optional)

GEOTYP =10: g1 = Length of edge along X′ axis
g2 = Length of edge along Y′ axis

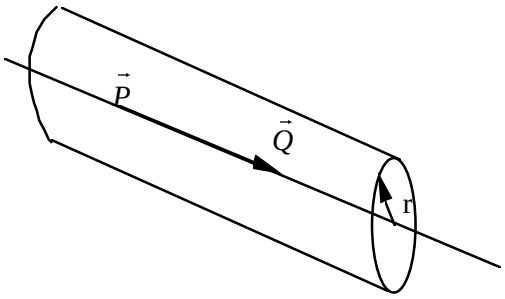
GEOTYP=11: g1 =Load curve ID defining axisymmetric surface profile about Z′-axis



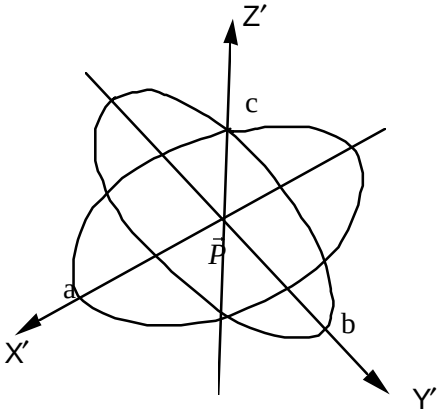
IGTYPE= 1: Infinite Plane



IGTYPE= 2: Sphere



IGTYPE= 3: Infinite Cylinder



$$\left(\frac{X'}{a}\right)^n + \left(\frac{Y'}{b}\right)^n + \left(\frac{Z'}{c}\right)^n = 1$$

IGTYPE= 4: Hyperellipsoid

Figure 5.3a. Contact Entities.

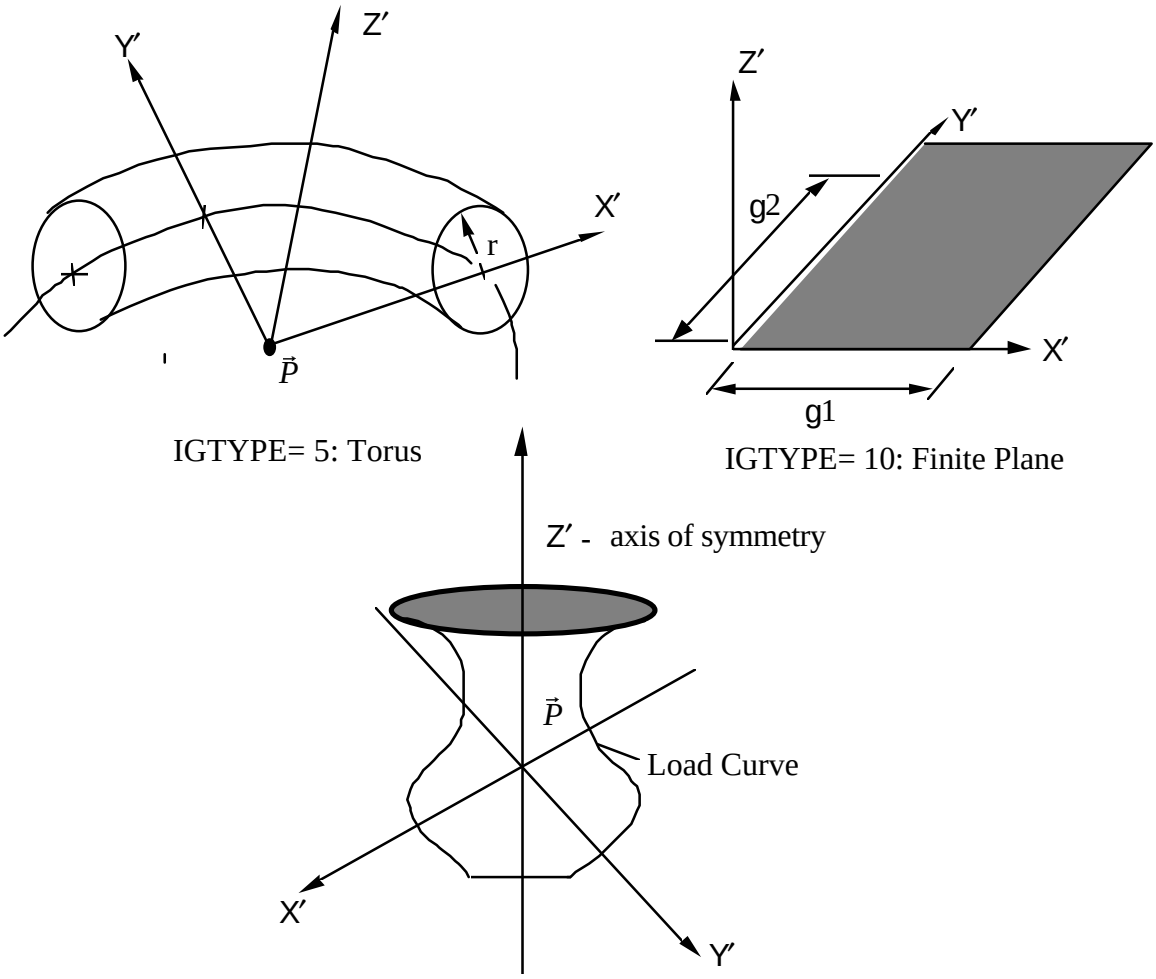


Figure 5.3b. Contact Entities.

***CONTACT_1D**

Purpose: Define one-dimensional slide lines for rebar in concrete.

Card Format

Card 1	1	2	3	4	5	6	7	8
Variable	NSIDS	NSIDM	ERR	SIGC	GB	SMAX	EXP	
Type	I	I	F	F	F	F	F	
Default	none	none	0.	0.	0.	0.	0.	

VARIABLE	DESCRIPTION
NSIDS	Nodal set ID for the slave nodes, see *SET_NODE.
NSIDM	Nodal set ID for the master nodes, see *SET_NODE.
ERR	External radius of rebar
SIGC	Compressive strength of concrete
GB	Bond shear modulus
SMAX	Maximum shear strain displacement
EXP	Exponent in damage curve

With this option the concrete is defined with solid elements and the rebar with truss elements, each with their own unique set of nodal points. A string of nodes, called slave nodes, related to the truss elements may slide along a string of nodes, called master nodes, related to the solid elements. The sliding commences after the rebar debonds.

The bond between the rebar and concrete is assumed to be elastic perfectly plastic. The maximum allowable slip strain is given as:

$$u_{max} = SMAX \cdot e^{EXP \cdot D}$$

where D is the damage parameter $D_{n+1} = D_n + \Delta u$. The shear force at time $n+1$ is given as:

$$f_{n+1} = min\left(f_n + GB \cdot \pi \cdot ERR^2 \cdot \Delta u, G \cdot \pi \cdot ERR^2 \cdot u_{max} \right)$$

*CONTROL

The keyword control cards are optional and can be used to change defaults; however, it is advisable to define the *CONTROL_TERMINATION card. The keyword control cards in this section are defined in alphabetical order:

*CONTROL_ADAPTIVE
*CONTROL_ALE
*CONTROL_BULK_VISCOSITY
*CONTROL_CONTACT
*CONTROL_COUPLING
*CONTROL_CPU
*CONTROL_DYNAMIC_RELAXATION
*CONTROL_ENERGY
*CONTROL_HOURGLASS
*CONTROL_OUTPUT
*CONTROL_PARALLEL
*CONTROL_SHELL
*CONTROL_SOLUTION
*CONTROL_STRUCTURED
*CONTROL_SUBCYCLE
*CONTROL_TERMINATION
*CONTROL_THERMAL_NONLINEAR
*CONTROL_THERMAL_SOLVER
*CONTROL_THERMAL_TIMESTEP
*CONTROL_TIMESTEP

The ordering of the control cards in the input file is competely arbitrary. To avoid ambiguities, define no more than one control card of each type.

*CONTROL_ADAPTIVE

Purpose: Activate adaptive meshing. The parts which are adaptively meshed are defined by *PART.

Card Format

	1	2	3	4	5	6	7	8
Variable	ADPFREQ	ADPTOL	ADPOPT	MAXLVL	TBIRTH	TDEATH	LCADP	IOFLAG
Type	F	F	I	I	F	F	I	I
Default	none	none	1	3	0.0	none	0	0

VARIABLE	DESCRIPTION
ADPFREQ	Time interval between adaptive refinements, see Figure 6.1.
ADPTOL	Adaptive error tolerance in degrees, see also option ADPOPT below.
ADPOPT	Adaptive options: EQ.1: angle change in degrees per adaptive refinement relative to the surrounding elements for each element to be refined. EQ.2: total angle change in degrees relative to the surrounding element for each element to be refined. For example, if the adptol=5 degrees, the element will be refined to the second level when the total angle change reaches 5 degrees. When the angle change is 10 degrees the element will be refined to the third level.
MAXLVL	Maximum number of refinement levels. Values of 1, 2, 3, ... allow a maximum of 4, 16, 64, ... elements, respectively, to be created for each original element.
TBIRTH	Birth time at which the adaptive remeshing begins, see Figure 6.1.
TDEATH	Death time at which the adaptive remeshing ends, see Figure 6.1.
LCADP	Adaptive interval is a function of time given by load curve ID, LCADP. If this option is nonzero, the adpfreq will be replaced by LCADP.
IOFLAG	Flag to generate adaptive mesh at exit including *NODE, *ELEMENT, *SHELL, and *BOUNDARY_, *CONTACT_NODE_, and *CONSTRAINED_ADAPTIVITY, to be saved in the file, *adapt.msh*.

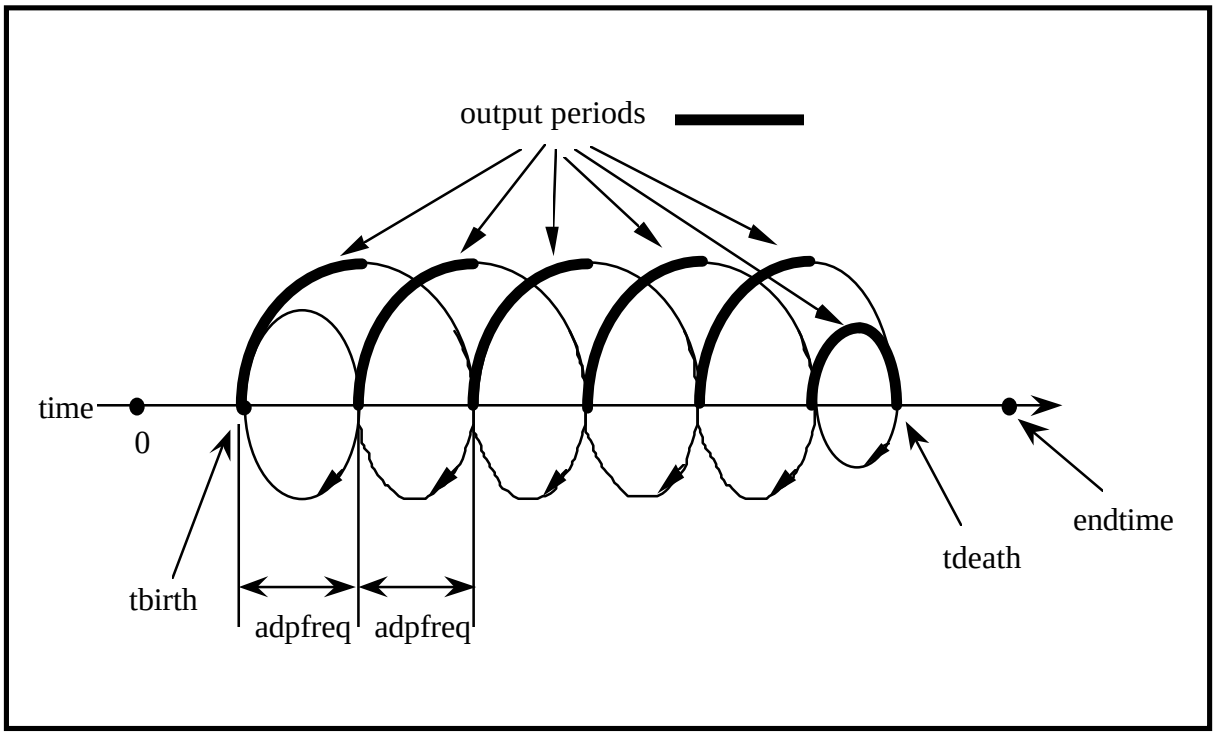


Figure 6.1. At time=*tbirth* the adaptive calculation begins. After computing for a time interval *adpfreq* error norms are computed and the mesh that existed at time=*tbirth* is refined based on the computed error norms. With the new mesh, the calculation continues to time=*tbirth+2 x adpfreq* where the error norms are again computed. The mesh that existed at time=*tbirth+adpfreq* is refined and the calculation continues to time=*tbirth+3 x adpfreq*, and so on.

*CONTROL_ALE

Purpose: Set default control parameters for the Arbitrary Lagrange-Eulerian calculations. See also *SECTION_SOLID_ALE and *_ALE_SMOOTHING.

Card Format

Card 1

Variable	DCT	NADV	METH	AFAC	BFAC	CFAC	DFAC	
Type	I	I	I	F	F	F	F	
Default	1	0	3	none	none	none	0	

Card 2

Variable	START	END	AAFAC					
Type	F	F	F					
Default	0	$1. \times 10^{20}$	1					

VARIABLE	DESCRIPTION
DCT	Default continuum treatment: EQ.1: Lagrangian (default), EQ.2: Eulerian, EQ.3: Arbitrary Lagrangian Eulerian, EQ.4: Eulerian Ambient.
NADV	Number of cycles between advections.
METH	Advection method: EQ.1: donor cell (first order accuracte), EQ.2: Van Leer + half index shift, EQ.3: Van Leer (default).
AFAC	Smoothing weight factor - Simple average: EQ.-1: turn smoothing off.

VARIABLE	DESCRIPTION
BFAC	Smoothing weight factor – Volume weighting
CFAC	Smoothing weight factor – Isoparametric
DFAC	Smoothing weight factor – Equipotential
START	Start time for smoothing
END	End time for smoothing
AAFAC	ALE advection factor

See also *SECTION_SOLID_OPTION.

*CONTROL_BULK_VISCOSITY

Purpose: Reset the default values of the bulk viscosity coefficients globally. This may be advisable for shock wave propagation and some materials.

Card Format

	1	2	3	4	5	6	7	8
Variable	Q2	Q1						
Type	F	F						
Default	1.5	.06						

VARIABLE	DESCRIPTION
Q2	Default quadratic viscosity coefficient.
Q1	Default linear viscosity coefficient.

See also Chapter 18 in Theoretical Manual.

*CONTROL_CONTACT

Purpose: Change defaults for computation with contact surfaces.

Card Format

Card 1	1	2	3	4	5	6	7	8
Variable	SLSFAC	RWPNAL	ISLCHK	SHLTHK	PENOPT	THKCHG	ORIEN	
Type	F	F	I	I	I	I	I	
Default	.1	none	2	0	1	1	1	

Card 2	1	2	3	4	5	6	7	8
Variable	USRSTR	USRFRC	NSBCS	INTERM	XPENE	SSTHK	ECDT	TIEDPRJ
Type	I	I	I	I	F	I	I	I
Default	0	0	10	0	4.0	0	0	0

VARIABLE	DESCRIPTION
SLSFAC	Scale factor for sliding interface penalties, SLSFAC: EQ.0: default = .1.
RWPNAL	Scale factor for rigid wall penalties for treating rigid bodies interacting with fixed rigid walls, RWPNAL. The penalties are set so that a scale factor of unity should be optimal; however, this may be very problem dependent. If rigid/deformable materials switching is used, this option should be used if the switched materials are interacting with rigid walls. EQ.0.0: rigid bodies interacting with rigid walls are not considered. GT.0.0: rigid bodies interact with fixed rigid walls. A value of 1.0 is recommended. Seven (7) variables are stored for each slave node. This can increase memory requirements significantly if all nodes are slaved to the rigid walls.

VARIABLE	DESCRIPTION
ISLCHK	Initial penetration check in contact surfaces with indication of initial penetration in output file, ISLCHK: EQ.0: the default is set to 2, EQ.1: no checking, EQ.2: full check of initial penetration is performed.
SHLTHK	Shell thickness considered in type surface to surface and node to surface type contact options, where options 1 and 2 below activate the new contact algorithms. The thickness offsets are always included in single surface, constraint method, and automatic surface to surface and node to surface contact types (See comments below.): EQ.0: thickness is not considered, EQ.1: thickness is considered but rigid bodies are excluded, EQ.2: thickness is considered including rigid bodies.
PENOPT	Penalty stiffness value option. For default calculation of the penalty value please refer to the Theoretical Manual. EQ.0: the default is set to 1, EQ.1: minimum of master segment and slave node (default), EQ.2: use master segment stiffness (old way), EQ.3: use slave node value, EQ.4: use slave node value, area or mass weighted, EQ.5: same as 4 but inversely proportional to the shell thickness. This may require special scaling and is not generally recommended. Options 4 and 5 are recommended for metalforming calculations.
THKCHG	Shell thickness changes considered in single surface contact: EQ.1: no consideration (default), EQ.2: shell thickness changes are included.
ORIEN	Optional automatic reorientation of contact interface segments during initialization: EQ.0: default is set to 1. EQ.1: active for automated (part) input only. Contact surfaces are given by *PART definitions. EQ.2: active for manual (segment) and automated (part) input. EQ.3: inactive.
USRSTR	Storage per contact interface for user supplied interface control subroutine, see Appendix D. If zero, no input data is read and no interface storage is permitted in the user subroutine. This storage should be large enough to accommodate input parameters and any history data. This input data is available in the user supplied subroutine.

VARIABLE	DESCRIPTION
USRFRC	Storage per contact interface for user supplied interface friction subroutine, see Appendix E. If zero, no input data is read and no interface storage is permitted in the user subroutine. This storage should be large enough to accommodate input parameters and any history data. This input data is available in the user supplied subroutine.
NSBCS	Number of cycles between contact searching using three dimensional bucket searches. Defaults recommended.
INTERM	Flag for intermittent searching in old surface to surface contact using the interval specified as NSBCS above: EQ.0: off, EQ.1: on.
XPENE	Contact surface maximum penetration check multiplier. If the small penetration checking option, PENCHK, on the contact surface control card is active, then nodes whose penetration then exceeds the product of XPENE and the element thickness are set free, see *CONTROL_OPTION_...: EQ.0: default is set to 4.0.
SSTHK	Flag for using actual shell thickness in single surface contact logic-types 4, 13, and 15. See comments below. EQ.0: logic is enabled (default), EQ.1: logic is skipped (sometimes recommended for metal forming calculations).
ECDT	Time step size override for eroding contact: EQ.0: contact time size may control Dt. EQ.1: contact is not considered in Dt determination.
TIEDPRJ	Bypass projection of slave nodes to master surface in types: *CONTACT_TIED_NODES_TO_SURFACE, *CONTACT_TIED_SHELL_EDGE_TO_SURFACE, and, *CONTACT_TIED_SURFACE_TO_SURFACE tied interface options: EQ.0: eliminate gaps by projection nodes, EQ.1: bypass projection. Gaps create rotational constraints which can substantially affect results.

The shell thickness change option must be activated in CONTROL_SHELL control input (see ISTUPD) and a nonzero flag specified for SHLTHK above before the shell thickness changes can be included in the surface to surface contact types. An additional flag must be set, see THKCHG above, if thickness changes are included in the single surface contact algorithms. The new contact algorithms that include the shell thickness are relatively recent and are now fully

optimized and parallelized. The searching in the new algorithms is considerably more extensive and therefore slightly more expensive.

In the single surface contacts types SINGLE_SURFACE, AUTOMATIC_SINGLE_ SURFACE, and ERODING_SINGLE_SURFACE, the default contact thickness is taken as the smaller value of the shell thickness or the shell edge lengths between shell nodes 1-2, 2-3, and 4-1. This may create unexpected difficulties if it is the intent to include thickness effects when the in-plane shell element dimensions are less than the thickness. The default is based on years of experience where it has been observed that sometimes rather large nonphysical thicknesses are specified to achieve high stiffness values. Since the global searching algorithm includes the effects of shell thicknesses, it is possible to slow the searches down considerably by using such nonphysical thickness dimensions.

***CONTROL_COUPLING**

Purpose: Change defaults for MADYMO3D/CAL3D coupling, see Appendix F.

Card Format

Card 1	1	2	3	4	5	6	7	8
Variable	UNLENG	UNTIME	UNFORC	TIMIDL	FLIPX	FLIPY	FLIPZ	SUBCYL
Type	F	F	F	F	I	I	I	I
Default	1.	1.	1.	0.	0	0	0	1

VARIABLE	DESCRIPTION
UNLENG	Unit conversion factor for length. MADYMO3D/GM-CAL3D lengths are multiplied by UNLENG to obtain LS-DYNA3D lengths.
UNTIME	Unit conversion factor for time, UNTIME. MADYMO3D/GM-CAL3D time is multiplied by UTIME to obtain LS-DYNA3D time.
UNFORC	Unit conversion factor for force, UNFORC. MADYMO3D/GM-CAL3D force is multiplied by UNFORC to obtain LS-DYNA3D force.
TIMIDL	Idle time during which CAL3D or MADYMO is computing and LS-DYNA3D remains inactive. Important for saving computer time.
FLIPX	Flag for flipping X-coordinate of CAL3D/MADYMO3D relative to the LS-DYNA3D model: EQ.0: off, EQ.1: on.
FLIPY	Flag for flipping Y-coordinate of CAL3D/MADYMO3D relative to the LS-DYNA3D model: EQ.0: off, EQ.1: on.
FLIPZ	Flag for flipping Z-coordinate of CAL3D/MADYMO3D relative to the LS-DYNA3D model: EQ.0: off, EQ.1: on.

VARIABLE	DESCRIPTION
SUBCYL	CAL3D/MADYMO3D subcycling interval (# of cycles): EQ.0: Set to 1, EQ.n: number of LS-DYNA3D time steps between each CAL3D/MADYMO3D step. Then the position of the contacting rigid bodies is assumed to be constant for n LS-DYNA3D time steps. This may result in some increase in the spikes in contact, thus this option should be used carefully. As the CAL3D/MADYMO3D programs usually work with a very small number of degrees of freedom, not much gain in efficiency can be achieved.

***CONTROL_CPU**

Purpose: Control cpu time.

Card Format

	1	2	3	4	5	6	7	8
Variable	CPUTIM							
Type	F							

VARIABLE	DESCRIPTION
CPUTIM	Seconds of cpu time: EQ:0.0 no cpu time limit set

The CPU time limit applies to the current phase of the analysis or restart. The limit is not checked until after the initialization stage of the calculation. Upon reaching the cpu limit, the code will output a restart dump file and terminate.

The CPU limit can also be specified on the input control line to LS-DYNA3D. If a value is specified on both the control line and in the input deck, the minimum value will be used.

*CONTROL_DYNAMIC_RELAXATION

Purpose: Define controls for dynamic relaxation. Important for stress initialization.

Card Format

	1	2	3	4	5	6	7	8
Variable	NRCYCK	DRTOL	DRFCTR	DRTERM	TSSFDR	IRELAL	EDTTL	IDRFLG
Type	I	F	F	F	F	I	F	I
Default	250	0.001	0.995	infinity	TSSFAC	0	0.04	0
Remarks								1, 2

VARIABLE	DESCRIPTION
NRCYCK	Number of iterations between convergence checks, for dynamic relaxation option (default = 250).
DRTOL	Convergence tolerance for dynamic relaxation option (default = 0.001).
DRFCTR	Dynamic relaxation factor (default = .995).
DRTERM	Optional termination time for dynamic relaxation. Termination occurs at this time or when convergence is attained (default = infinity).
TSSFDR	Scale factor for computed time step during dynamic relaxation. If zero, the value is set to SCRT defined on *CONTROL_TIMESTEP. After converging, the scale factor is reset to SCRT.
IRELAL	Automatic control for dynamic relaxation option based on algorithm of Papadrakakis [Papadrakakis 1981]: EQ.0: not active, EQ.1: active.
EDTTL	Convergence tolerance on automatic control of dynamic relaxation.
IDRFLG	Dynamic relaxation flag for stress initialization: EQ.-1: dynamic relaxation is activated and time history output is produced during dynamic relaxation, see note 2 below, EQ.0: not active, EQ.1: dynamic relaxation is activated, EQ.2: initialization to a prescribed geometry.

Remark:

1. Stress initialization in LS-DYNA3D for small strains may be accomplished by linking to an implicit code (option 2). A displacement state is required that gives for each nodal point its label, xyz displacements, and xyz rotations and temperature. This data is read from unit 7 (m=) with the format (i8,6e15.0). See also INTRODUCTION, Execution Syntax.

2. If IDRFLG is set to -1 the dynamic relaxation proceeds as normal but time history data is written to the D3THDT file. At the end of dynamic relaxation, the problem time is reset to zero. However, information is written to the D3THDT file with an increment to the time value. The time increment used is reported at the end of dynamic relaxation.

*CONTROL_ENERGY

Purpose: Provide controls for energy dissipation options.

Card Format

	1	2	3	4	5	6	7	8
Variable	HGEN	RWEN	SLNTEN	RYLEN				
Type	I	I	I	I				
Default	1	2	1	1				

VARIABLE	DESCRIPTION
HGEN	Hourglass energy calculation option. This option requires significant additional storage and increases cost by ten percent: EQ.1: hourglass energy is not computed (default), EQ.2: hourglass energy is computed and included in the energy balance. The hourglass energies are reported in the ASCII files GLSTAT and MATSUM, see *DATABASE_OPTION.
RWEN	Stonewall energy dissipation option: EQ.1: energy dissipation is not computed, EQ.2: energy dissipation is computed and included in the energy balance (default). The stonewall energy dissipation is reported in the ASCII file GLSTAT, see *DATABASE_OPTION.
SLNTEN	Sliding interface energy dissipation option: EQ.1: energy dissipation is not computed (default), EQ.2: energy dissipation is computed and included in the energy balance. The sliding interface energy is reported in ASCII files GLSTAT and SLEOUT, see *DATABASE_OPTION.
RYLEN	Rayleigh energy dissipation option (damping eneryg dissipation): EQ.1: energy dissipation is not computed (default), EQ.2: energy dissipation is computed and included in the energy balance. The damping energy is reported in ASCII file GLSTAT, see *DATABASE_OPTION.

***CONTROL_HOURGLASS**

Purpose: Reset the default values of the hourglass control.

Card Format

	1	2	3	4	5	6	7	8
Variable	IHQ	QH						
Type	I	F						
Default	1	0.1						
Remarks	1							

VARIABLE	DESCRIPTION
IHQ	Default hourglass viscosity type: EQ.1: standard LS-DYNA3D, EQ.2: Flanagan-Belytschko integration, EQ.3: Flanagan-Belytschko with exact volume integration, EQ.4: stiffness form of type 2 (Flanagan-Belytschko), EQ.5: stiffness form of type 3 (Flanagan-Belytschko). In the shell elements, IHQ < 4 is the viscous form based on Belytschko-Tsay. If IHQ = 4 or 5, the stiffness form is obtained. The stiffness forms, however, can stiffen the response, especially if the deformations are large, and therefore should be used with care. For high velocities the viscous forms are recommeded and for low velocities the stiffness forms are recommended. For large deformations and nonregular solids, option 3 or 5 is recommended.
QH	Default hourglass coefficient, QH. Values of QH that exceed .15 may cause instabilities. The recommended default applies to all options.

Remark:

1. Hourglass coefficients and type can be set by part ID in the ***HOURGLASS** Section.

*CONTROL

*CONTROL_OUTPUT

Purpose: Set output display parameters.

Card Format

	1	2	3	4	5	6	7	8
Variable	NPOPT	NEECHO	NREFUP	IACCOP	OPIFS	IPNINT	IKEDIT	
Type	I	I	I	I	F	I	I	
Default	0	0	0	0	0.	0	100	

VARIABLE	DESCRIPTION
NPOPT	Print suppression during input phase flag for the printed output file: EQ.0: no suppression, EQ.1: nodal coordinates, element connectivities, rigid wall definitions and initial velocities are not printed.
NEECHO	Print suppression during input phase flag for echo file: EQ.0: all data printed, EQ.1: nodal printing is suppressed, EQ.2: element printing is suppressed, EQ.3: both node and element printing is suppressed.
NREFUP	Flag to update reference node coordinates for beam elements. **This option requires that each reference node is unique to the beam:** EQ.0: no update, EQ.1: update. This is generally recommended.
IACCOP	Averaged accelerations from velocities in file ÒnodoutÓ and the time history database file Òd3thdtÓ: EQ.0: no average (default), EQ.1: averaged between output intervals.
OPIFS	Output interval for interface file (*Æt), see INTRODUCTION, Execution syntax.
IPNINT	Print initial time step sizes for all elements on the first cycle: EQ.0: no printout, EQ.1: the governing time step sizes for each element are printed.
IKEDIT	Problem status report interval steps to the D3HSP (printed output) file.

***CONTROL_PARALLEL**

Purpose: Control parallel processing usage for shared memory computers.

Card Format

	1	2	3	4	5	6	7	8
Variable	NCPU	NUMRHS	ACCU					
Type	I	I	I					
Default	1	0	1					

VARIABLE	DESCRIPTION
NCPU	Number of cpus used.
NUMRHS	Number of right-hand sides written: EQ.0: same as NCPU, EQ.1: write one only .
ACCU	Accuracy flag for parallel solution (NCPU >1) EQ.1: on (default) EQ.2: off, for a faster solution.

It is recommended to always set NUMRHS=NCPU since great improvements in the parallel performance are obtained. Setting NUMRHS to one reduces storage by one right hand side vector for each processor. The accuracy flag, ACCU, provides for identical results (or nearly so) whether one, two, or more processors are used.

*CONTROL_SHELL

Purpose: Provide controls for computing shell response.

Card Format

	1	2	3	4	5	6	7	8
Variable	WRPANG	ITRIST	IRNXX	ISTUPD	THEORY	BWC	MITER	
Type	F	I	I	I	I	I	I	
Default	20.	2	-1	0	2	2		

VARIABLE	DESCRIPTION
WRPANG	Shell element warpage angle in degrees. If a warpage greater than this angle is found, a warning message is printed. Default is 20 degrees.
ITRIST	Automatic sorting of triangular shell elements to treat degenerate quadrilateral shell elements as C0 triangular shells, see option THEORY below: EQ.1: full sorting, EQ.2: no sorting required (default).
IRNXX	Hughes-Liu shell normal update option: EQ.-2: unique nodal fibers. Good to model shell edges. EQ.-1: compute normals each cycle (recommended), EQ.0: default set to -1, EQ.1: compute on restarts, EQ.n: compute every n cycles.
ISTUPD	Shell thickness change option: EQ.0: no change. EQ.1: membrane straining causes thickness change. Important in sheetmetalforming.

VARIABLE	DESCRIPTION
THEORY	Shell theory: EQ.1: Hughes-Liu, EQ.2: Belytschko-Tsay (default), EQ.3: BCIZ triangular shell (not recommended), EQ.4: C_o triangular shell, EQ.5: Belytschko-Tsay membrane, EQ.6: S/R Hughes Liu, EQ.7: S/R co-rotational Hughes Liu, EQ.8: Englemann-Whirley shell, EQ.9: fully integrated Belytschko-Tsay membrane, EQ.10: Belytschko-Wong-Chiang (recommended), EQ.11: Fast (co-rotational) Hughes-Liu.
BWC	Warping stiffness for Belytschko-Tsay shells: EQ.1: Belytschko-Wong-Chiang warping stiffness added (recommended), EQ.2: Belytschko-Tsay (default).
MITER	Plane stress plasticity option (applies to materials 3, 18, 19, and 24): EQ.1: iterative plasticity with 3 secant iterations (default), EQ.2: full iterative plasticity, EQ.3: radial return noniterative plasticity. May lead to false results and has to be used with great care.

*CONTROL_SOLUTION

Purpose: To specify the analysis solution procedure if thermal only or coupled thermal analysis is performed.

Card Format

	1	2	3	4	5	6	7	8
Variable	SOLN							
Type	I							
Default	0							

VARIABLE	DESCRIPTION
SOLN	Analysis solution procedure: 0: Structural analysis only, 1: Thermal analysis only, 2: Coupled structural thermal analysis.

***CONTROL_STRUCTURED**

Purpose: Write out a LS-DYNA3D structured input deck for Version 930. This input deck will not support all capabilities that are available in Version 930. As a result some data such as load curve numbers will be output in an internal numbering system.

*CONTROL_SUBCYCLE

Purpose: Control time step subcycling. This feature is described in the LS-DYNA3D Theoretical Manual, Section 20.2. May be detrimental in cases of vectorized computation. This keyword activates subcycling.

***CONTROL_TERMINATION**

Purpose: Stop the job.

Card Format

	1	2	3	4	5	6	7	8
Variable	ENDTIM	ENDCYC	DTMIN	ENDENG	ENDMAS			
Type	F	I	F	F	F			
Default	0.0	0	0.0	0.0	0.0			
Remarks	1		2					

VARIABLE	DESCRIPTION
ENDTIM	Termination time. Mandatory.
ENDCYC	Termination cycle. The termination cycle is optional and will be used if the specified cycle is reached before the termination time. Cycle number is identical with the time step number.
DTMIN	Reduction (or scale) factor for initial time step size to determine minimum time step, TSMIN. TSMIN=DTSTART*DTMIN where DTSTART is the initial step size determined by LS-DYNA3D. When TSMIN is reached, LS-DYNA3D terminates with a restart dump.
ENDENG	Percent change in energy ratio for termination of calculation. If undefined, this option is inactive.
ENDMAS	Percent change in the total mass for termination of calculation. This option is relevant if and only if mass scaling is used to limit the minimum time step size, see *CONTROL_TIMESTEP variable name ÒDT2MSÓ.

Remark:

1. Termination by displacement may be defined in the *TERMINATION section.

2. If the erosion flag on *CONTROL_TIMESTEP is set (ERODE=1), then the shell elements and solid elements with time steps falling below DTMIN will be eroded.

*CONTROL_THERMAL_NONLINEAR

Purpose: Set parameters for a nonlinear thermal or coupled structural/thermal analysis. The control card, *CONTROL_SOLUTION, is also required.

Card Format

	1	2	3	4	5	6	7	8
Variable	REFMAX	TOL						
Type	I	F						
Default	10	-						

VARIABLE	DESCRIPTION
REFMAX	Maximum number of matrix reformations per time step: EQ.0: set to 10 reformations.
TOL	Convergence tolerance for temperature: EQ.0.0: set to 1000 * machine roundoff.
DCP	Divergence control parameter: steady state problems 0.3 _ DCP _ 1.0 default 1.0 transient problems 0.0 < DCP _ 1.0 default 0.5

*CONTROL_THERMAL_SOLVER

Purpose: Set options for the thermal solution in a thermal only or coupled structural-thermal analysis. The control card, *CONTROL_SOLUTION, is also required.

Card Format

	1	2	3	4	5	6	7	8
Variable	ATYPE	PTYPE	SOLVER	CGTOL	GPT			
Type	I	I	I	F	I			
Default	0	0	3	1.0e-04	8			

VARIABLE	DESCRIPTION
ATYPE	Thermal analysis type: EQ.0: Steady state analysis, EQ.1: transient analysis.
PTYPE	Thermal problem type: (see *CONTROL_THERMAL_NONLINEAR if no-zero) EQ.0: linear problem, EQ.1: nonlinear problem with material properties evaluated at gauss point temperature. EQ.2: nonlinear problem with material properties evaluated at element average temperature.
SOLVER	Thermal analysis solver type: EQ.1: actol : symmetric direct solver, EQ.2: dactol : nonsymmetric direct solver, EQ.3: dscg : diagonal scaled conjugate gradient iterative (default), EQ.4: iccg : incomplete choleski conjugate gradient iterative.
CGTOL	Convergence tolerance for iterative solver types 3 and 4: EQ.0.0: set to 1.0e-04.
GPT	Number of Gauss points to be used in the solid elements: EQ.0: the default is set to 8, EQ.1: one point quadrature is used.

Remark:

1. Use of a direct solver (SOLVER, e.g., 1 or 2) is mostly less efficient than an iterative solver. If convergence problems occur, option 4 should be tried first before a direct solver is used.

***CONTROL_THERMAL_TIMESTEP**

Purpose: Set timestep controls for the thermal solution in a thermal only or coupled structural/ thermal analysis. Also *CONTROL_SOLUTION, *CONTROL_THERMAL_SOLVER needed.

Card Format

	1	2	3	4	5	6	7	8
Variable	TS	TIP	ITS	TMIN	TMAX	DTEMP	TSCP	
Type	I	F	F	F	F	F	F	
Default	0	0.5	none	-	-	1.0	0.5	

VARIABLE	DESCRIPTION
TS	Time step control: EQ.0: fixed time step, EQ.1: variable time step (may increase or decrease).
TIP	Time integration parameter: EQ.0.0: set to 0.5 - Crank-Nicholson scheme, EQ 1.0: fully implicit.
ITS	Initial thermal time step
TMIN	Minimum thermal time step: EQ.0.0: set to structural explicit timestep.
TMAX	Maximum thermal time step: EQ.0.0: set to 100 * structural explicit timestep.
DTEMP	Maximum temperature change in each time step above which the thermal timestep will be decreased: EQ.0.0: set to a temperature change of 1.0.
TSCP	Time step control parameter. The thermal time step is decreased by this factor if convergence is not obtained. 0. < TSCP < 1.0: EQ.0.0: set to a factor of 0.5.

*CONTROL_TIMESTEP

Purpose: Set structural time step size control using different options.

Card Format

	1	2	3	4	5	6	7	8
Variable	DTINIT	TSSFAC	ISDO	TSLIMT	DT2MS	LCTM	ERODE	MS1ST
Type	F	F	I	F	F	I	I	I
Default	-	0.9/0.67	0	0.0	0.0	0	0	0

VARIABLE	DESCRIPTION
DTINIT	Initial time step size: EQ.0.0: LS-DYNA3D determines initial step size.
TSSFAC	Scale factor for computed time step (old name SCFT). (Default = .90; if high explosives are used, the default is lowered to .67).
ISDO	Basis of time size calculation for 4-node shell elements. 3-node shells use the shortest altitude for options 0,1 and the shortest side for option 2. This option has no relevance to solid elements, which use a length based on the element volume divided by the largest surface area. EQ.0: characteristic length=area/(longest side). EQ.1: characteristic length=area/(longest diagonal). EQ.2: based on bar wave speed and MAX **[**shortest side, area/longest side**]**. THIS LAST OPTION CAN GIVE A MUCH LARGER TIME STEP SIZE THAT CAN LEAD TO INSTABILITIES IN SOME APPLICATIONS, ESPECIALLY WHEN TRIANGULAR ELEMENTS ARE USED.
TSLIMT	Shell element minimum time step assignment, TSLIMT. When a shell controls the time step, element material properties (moduli not masses) will be modified such that the time step does not fall below the assigned step size. Applicable only to shell elements using material models *MAT_PLASTIC_KINEMATIC, *MAT_PONER_LAW_PLASTICITY, *MAT_STRAIN_RATE_DEPENDENT_PLASTICITY, *MAT_PIECE-WISE_LINEAR_PLASTICITY. The DT2MS option below applies to all materials and element classes and may be preferred.

VARIABLE	DESCRIPTION
DT2MS	Time step size for mass scaled solutions, DT2MS. Positive values are for quasi-static analyses or time history analyses where the inertial effects are insignificant. Default = 0.0. If negative, TSSFAC*|DT2MS| is the minimum time step size permitted and mass scaling is done if and only if it is necessary to meet the Courant time step size criterion. This latter option can be used in transient analyses if the mass increases remain insignificant. See *CONTROL_TERMINATION variable name ÒENDMASÓ.
LCTM	Load curve ID that limits the maximum time step size (optional).
ERODE	Erosion flag for solid and solid shell elements when DTMIN (see *CONTROL_TERMINATION)is reached. If this flag is not set the calculation will terminate: EQ.0: no, EQ.1: yes.
MS1ST	Limit mass scaling to the first step and fix the mass vector according to the time steps once. The time step will not be fixed but may drop during the calculation from the specified minimum: EQ.0: no, EQ.1: yes.

*DAMPING

The Keyword options in this section in alphabetical order are:

*DAMPING_GLOBAL
*DAMPING_PART_MASS
*DAMPING_PART_STIFFNESS

*DAMPING_GLOBAL

Purpose: Define mass weighted nodal damping that applies globally to the nodes of deformable bodies.

Card Format

	1	2	3	4	5	6	7	8
Variable	LCID	VALDMP						
Type	I	F						
Default	0	0.0						

VARIABLE	DESCRIPTION
LCID	Load curve ID which specifies node system damping: EQ.0: a contact damping factor as defined by VALDMP is used, EQ.n: system damping is given by load curve n. The damping force applied to each node is f=-d(t) mv, where d(t) is defined by load curve n.
VALDMP	System damping constant, d (this option is bypassed if the load curve number defined above is nonzero).

*DAMPING

Remark:

1. This keyword is also used for the restart, see *RESTART.
   With system damping the acceleration is computed as:

$$a^n = \boldsymbol{M}^{-1}\left(\boldsymbol{P}^n - \boldsymbol{F}^n - \boldsymbol{F}^n_{damp} \right)$$

where, $\boldsymbol{M}$ is the diagonal mass matrix, $\boldsymbol{P}^n$ is the external load vector, $\boldsymbol{F}^n$ is the internal load vector, and $\boldsymbol{F}^n_{damp}$ is the force vector due to system damping. This latter vector is defined as:

$$\boldsymbol{F}^n_{damp} = \boldsymbol{D}_s \boldsymbol{m} \boldsymbol{v}$$

The best damping constant for the system is usually based on the critical damping factor for the lowest frequency mode of interest. Therefore,

$$\boldsymbol{F}^n_{damp} = \boldsymbol{D}_s = 2\omega_{min}$$

is recommended where the natural frequency (given in radians per unit time) is generally taken as the fundamental frequency of the structure.

***DAMPING_PART_MASS**

Purpose: Define mass weighted damping by part set ID.

Card Format

	1	2	3	4	5	6	7	8
Variable	PID	LCID	SF					
Type	I	I	F					
Default	0	0	1.0					

VARIABLE	DESCRIPTION
PID	Part ID, see *PART and *SET_PART.
LCID	Load curve ID which specifies system damping for parts.
SF	Scale factor for load curve. This allows a simple modification of the load curve values.

Mass weighted damping damps all motions including rigid body motions. For oscillatory motion stiffness, weighted damping is preferred.

*DAMPING

*DAMPING_PART_STIFFNESS

Purpose: Assign Rayleigh stiffness damping coefficient by part set ID.

Card Format

	1	2	3	4	5	6	7	8
Variable	PID	BETA						
Type	I	F						
Default								

VARIABLE	DESCRIPTION
PID	Part ID, see *PART and *SET_PART.
BETA	Rayleigh damping coefficient for stiffness weighted damping.

The damping matrix in Rayleigh damping is defined as:

$$C = \alpha M + \beta K ,$$

where C, M, and K are the damping, mass, and stiffness matrices, respectively. The constants α. and β are the mass and stiffness proportional damping constants. The mass proportional damping can be treated by system damping in LS-DYNA3D, see Control Card 14, Columns 1-5. Transforming C with the ith eigenvector σ_i gives:

$$\phi_i^t C \phi_i = \phi_i^t (\alpha M + \beta K)\phi_i = \alpha + \beta\omega_i^2 = 2\omega_i \xi_i \delta_{ij}$$

where ω_i is the ith frequency (radians/unit time) and ξ_i is the corresponding modal damping parameter. If 10% of critical damping is sought in the ith mode using stiffness proportional damping then set:

$$\beta = \frac{.20}{\omega_i}$$

Generally, the stiffness proportional damping is effective for high frequencies and is orthogonal to rigid body motion. Mass proportional damping is more effective for low frequencies and will damp rigid body motion.

*DATABASE

The database defintions are optional, but are necessary to obtain output files containing results information. In this section the database keywords are defined in alphabetical order:

*DATABASE_OPTION
*DATABASE_BINARY_OPTION
*DATABASE_CROSS_SECTION_OPTION
*DATABASE_EXTENT_OPTION
*DATABASE_HISTORY_OPTION
*DATABASE_NODAL_FORCE_GROUP
*DATABASE_SPRING_FORWARD
*DATABASE_SUPERPLASTIC_FORMING
*DATABASE_TRACER

The ordering of the database definition cards in the input file is competely arbitrary.

*DATABASE_OPTION

Options for ASCII files include (if the file is not specified it will not be created):

SECFORC	Cross section forces. See *DATABASE_CROSS_SECTION_OPTION.
RWFORC	Wall forces.
NODOUT	Nodal point data. See also *DATABASE_HISTORY_OPTION.
ELOUT	Element data. See also *DATABASE_HISTORY_OPTION.
GLSTAT	Global data.
DEFORC	Discrete elements.
MATSUM	Material energies.
NCFORC	Nodal interface forces.
RCFORC	Resultant interface forces.
DEFGEO	Deformed geometry file
SPCFORC	SPC reaction forces.
SWFORC	Nodal constraint reaction forces (spotwelds and rivets).
ABSTAT	Airbag statistics.
NODFOR	Nodal force groups. See also *DATABASE_NODAL_FORCE_GROUP.
BNDOUT	Boundary condition forces and energy
RBDOUT	Rigid body data.
GCEOUT	Geometric contact entities.
SLEOUT	Sliding interface energy.
JNTFORC	Joint force file
SBTOUT	Seat belt output file
AVSFLT	AVS database. See also *DATABASE_EXTENT_OPTION.
MOVIE	MOVIE. See also *DATABASE_EXTENT_OPTION.
MPGS	MPGS. See also *DATABASE_EXTENT_OPTION.
TRHIST	Tracer particle history information. See also *DATABASE_TRACER.
TPRINT	Thermal output from a coupled structural/thermal or thermal only analysis.

Card Format

	1	2	3	4	5	6	7	8
Variable	DT	FLAG1	FLAG2					
Type	F	I	I					
Default	0.	-	-					

VARIABLE	DESCRIPTION
DT	Time interval between outputs. If DT is zero, no output is printed.
FLAG1..	Meaning depends on the file being written, see below.

The flags on the above card have the following meanings:

RBDOUT

FLAG1 Default option for writing to RBDOUT file:
EQ.0: print rigid body data into file (default),
EQ.n: do not print rigid body data into file.

Remark:

1. This keyword is also used in the restart phase, see *RESTART. Thus, the output interval can be changed when restarting.
2. All information in the files except in AVSFLT, MOVIE, AND MPGS can also be plotted using the post-processor LS-TAURUS. Arbitrary cross plotting of results between ASCII files is easily handled.

*DATABASE_BINARY_OPTION

Options for binary output files with the default names given include:

D3PLOT	Dt for complete output states. See also *DATABASE_EXTENT_BINARY.
D3THDT	Dt for time history data of element subsets. See *DATABASE_HISTORY.
D3DRLF	Dynamic relaxation database. Define output frequency in cycles.
D3DUMP	Binary output restart files. Define output frequency in cycles.
RUNRSF	Binary output restart file. Define output frequency in cycles.
INTFOR	Dt for output of contact interface data (file name must be given).
XTFILE	Flag to specify output of extra time history data to XTFILE at same time as D3THDT file. The following card is left blank for this option.

The D3DUMP and the RUNRSF options create complete databases which are necessary for restarts, see *RESTART. When RUNRSF is specified, the same file is overwritten after each interval. When D3DUMP is specified, a new restart file is created after each interval. When D3DUMP is specified, a new restart file is created after each interval, thus a “family” of files is created numbered sequentially D3DUMP01, D3DUMP02, etc. The default file names are RUNRSF and D3DUMP unless other names are specified on the execution line, see the INTRODUCTION, EXECUTION SYNTAX. Since all data held in memory is written into the restart files, these files can be quite large and care should be taken with the D3DUMP files not to create too many.

The D3PLOT, D3DRLF, and the INTFOR files contain plotting information to plot data over the three dimensional geometry of the model. These databases can be plotted with LS-TAURUS. The D3THDT file contains time history data for element subsets as well as global information, see *DATABASE_HISTORY. This data can be plotted with LS-TAURUS in Phase 2. The default names for the D3PLOT, D3DRLF, and the D3THDT files are D3PLOT, D3DRLF, and D3THDT. For INTFOR a unique name must be specified on the execution line with S=iff, (iff=file name), see the INTRODUCTION, EXECUTION SYNTAX. The file structure is such that each file contains the full geometry at the beginning, followed by the analysis generated output data at the specified time intervals. The default file size of 7000000 octal words may be much to small to hold one complete output state when models are very large, and an excessive number of files may be created. The limit of LS-DYNA3D to create files is 99 family members. Therefore, it is recommended that the file size be adjusted on the execution line with the X=scl (scl is a scale factor to enlarge the family member size). For the contents of the D3PLOT and D3THDT files see also the *DATABASE_EXTENT_BINARY definition. It is possible to severely restrict the information that is dumped and consequently reduce the size of the databases. The contents of the D3THDT file

are also specified with the *DATABASE_HISTORY definition. It should also be noted in particular that the databases can be considerably reduced for models with rigid bodies containing many elements.

Card Format

	1	2	3	4	5	6	7	8
Variable	DT/CYCL	LCDT	NOBEAM	NPLTC				
Type	F	I	I					

VARIABLE	DESCRIPTION
DT	Time interval between outputs.
CYCL	Output interval in time steps (a time step is a cycle). For the D3DRFL file a positive number will cause plot dumps to be written at the convergence check interval specified on the *CONTROL_DYNAMIC_RELAXATION card. If this file is not specified on the execution line command line (see INTRODUCTION, EXECUTION SYNTAX), it will not be created.
LCDT	Optional load curve ID specifying time interval between dumps. This option is only available for the D3PLOT, D3THDT and INTFOR files.
NOBEAM	Option flag for *DATABASE_BINARY_D3PLOT. If set to 1, the spring and damper discrete elements are not added to the D3PLOT database where they are displayed as beam elements. This option is useful when translating old LS-DYNA3D input decks to KEYWORD input. In older input decks there is no requirement that beam and spring elements have unique ID's, and beam elements may be created for the spring and dampers with identical ID's to existing beam elements causing a fatal error.
NPLTC	DT=ENDTIME/NPLTC applies to D3PLOT only. This overrides the DT specified in the first field.

DATABASE_CROSS_SECTION_*OPTION

Options include:

PLANE
SET

Purpose: Define a cross section for resultant forces written to ASCII file SECFORC. For the PLANE option, a set of two cards is required for each cross section. Then a cutting plane has to be defined, see Figure 8.1. If the SETS option is used, just one card is needed. In this latter case the forces in the elements belonging to the set are summed up to form the section forces.

Format (1 of 2) for the PLANE option

	1	2	3	4	5	6	7	8
Variable	PSID	XCT	YCT	ZCT	XCH	YCH	ZCH	
Type	I	F	F	F	F	F	F	
Default	0	0.	0.	0.	0.	0.	0.	

Format (2 of 2) for the PLANE option

	1	2	3	4	5	6	7	8
Variable	XHEV	YHEV	ZHEV	LENL	LENM			
Type	F	F	F	F	F			
Default	0.	0.	0.	infinity	infinity			

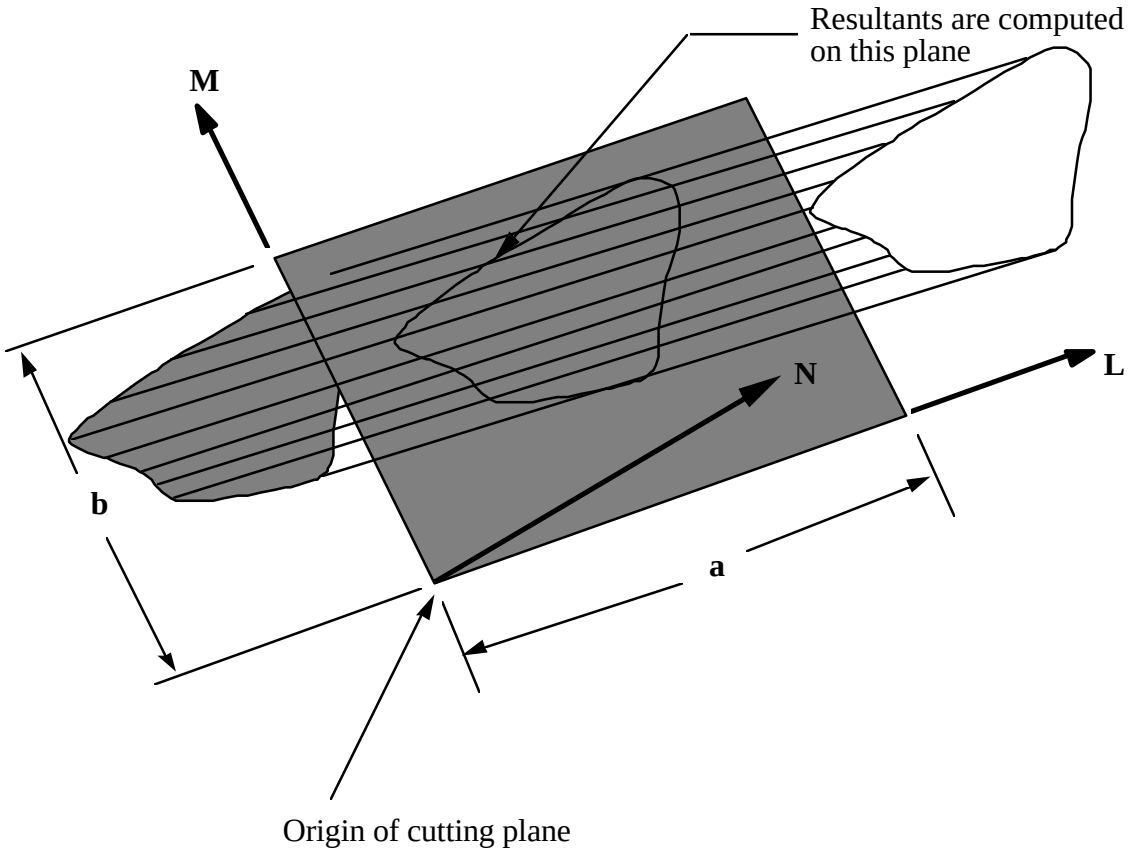


Figure 8.1. Definition of cutting plane for automatic definition of interface for cross-sectional forces. The automatic definition does not check for springs and dampers in the section. For best results the cutting plane should cleanly pass through the middle of the elements, distributing them equally on either side.

Format (1 of 1) for the SET option

	1	2	3	4	5	6	7	8
Variable	NSID	HSID	BSID	SSID	TSID	DSID		
Type	I	I	I	I	I	I		
Default	0	0	0	0	0	0		

VARIABLE	DESCRIPTION
PSID	Part set ID. If zero all parts are included.
XCT	x-coordinate of tail of any outward drawn normal vector, **N**, originating on wall (tail) and terminating in space (head), see Figure 8.1.
YCT	y-coordinate of tail of normal vector, **N**.
ZCT	z-coordinate of tail of normal vector, **N**.
XCH	x-coordinate of head of normal vector, **N**.
YCH	y-coordinate of head of normal vector, **N**.
ZCH	z-coordinate of head of normal vector, **N**.
XHEV	x-coordinate of head of edge vector, **L**.
YHEV	y-coordinate of head of edge vector, **L**.
ZHEV	z-coordinate of head of edge vector, **L**.
LENL	Length of edge a, in **L** direction.
LENM	Length of edge b, in **M** direction.
NSID	Nodal set ID, see *SET_NODE_OPTION.
HSID	Solid element set ID, see *SET_SOLID.
BSID	Beam element set ID, see *SET_BEAM.
SSID	Shell element set ID, see *SET_SHELL_OPTION.
TSID	Thick shell element set ID, see *SET_TSHELL.
DSID	Discrete element set ID, see *SET_DISCRETE.

***DATABASE_EXTENT_OPTION**

Options include:

AVS
MPGS
MOVIE
BINARY

Purpose: Specify output database to be written. Binary applies to the data written to the D3PLOT and D3THDT files. See *DATABASE_BINARY_OPTION.

For the AVS, MPGS, and MOVIE options the following cards apply:

Define as many cards as necessary. The created MPGS and MOVIE databases consist of a geometry file and one file for each output database.

Card Format

	1	2	3	4	5	6	7	8
Variable	VTYPE	COMP						
Type	I	I						

VARIABLE	DESCRIPTION
VTYPE	Variable type: EQ.0: node, EQ.1: brick, EQ.2: beam, EQ.3: shell, EQ.4: thick shell.
COMP	Component number. For the corresponding VTYPE, integer components from the following tables can be chosen: VTYPE.EQ.0: Table 8.1, VTYPE.EQ.1: Table 8.2, VTYPE.EQ.2: not supported, VTYPE.EQ.3: Table 8.3, VTYPE.EQ.4: not supported.

The AVS database consists of a title card, then a control card defining the number of nodes, brick-like elements, beam elements, shell elements, and the number of nodal vectors, NV, written for each output interval. The next NV lines consist of character strings that describe the nodal vectors. Nodal coordinates and element connectivities follow. For each state the solution time is written, followed by the data requested below. The last word in the file is the number of states. We recommend creating this file and examining its contents, since the organization is relatively transparent. The MOVIE and MPGS database are widely used and will be familiar with users who are currently using these databases.

Table 8.1. Nodal Quantities

Component Number	Quantity
1-3	x, y, z-displacements
4-6	x, y, z-velocities
7-9	x, y, z-accelerations
10	temperature

Table 8.2. Brick Element Quantities

Component Number	Quantity
1	x-stress
2	y-stress
3	z-stress
4	xy-stress
5	yz-stress
6	zx-stress
7	effective plastic strain

Table 8.3. Shell and Thick Shell Element Quantities

Component Number	Quantity
1	midsurface x-stress
2	midsurface y-stress
3	midsurface z-stress
4	midsurface xy-stress
5	midsurface yz-stress
6	midsurface xz-stress
7	midsurface effective plastic strain
8	inner surface x-stress
9	inner surface y-stress
10	inner surface z-stress
11	inner surface xy-stress
12	inner surface yz-stress
13	inner surface zx-stress
14	inner surface effective plastic strain
15	outer surface x-stress
16	outer surface y-stress
17	outer surface z-stress
18	outer surface xy-stress
19	outer surface yz-stress
20	outer surface zx-stress
21	outer surface effective plastic strain
22	bending moment-mxx (4-node shell)
23	bending moment-myy (4-node shell)
24	bending moment-mxy (4-node shell)
25	shear resultant-qxx (4-node shell)
26	shear resultant-qyy (4-node shell)
27	normal resultant-nxx (4-node shell)
28	normal resultant-nyy (4-node shell)
29	normal resultant-nxy (4-node shell)
30	thickness (4-node shell)

Table 8.3. Shell and Thick Shell Element Quantities (cont.).

Component Number	Quantity
31	element dependent variable
32	element dependent variable
33	inner surface x-strain
34	inner surface y-strain
35	inner surface z-strain
36	inner surface xy-strain
37	inner surface yz-strain
38	inner surface zx-strain
39	outer surface x-strain
40	outer surface y-strain
41	outer surface z-strain
42	outer surface xy-strain
43	outer surface yz-strain
44	outer surface zx-strain
45	internal energy
46	midsuface effective stress
47	inner surface effective stress
48	outer surface effective stress
49	midsurface max. principal strain
50	through thickness strain
51	midsurface min. principal strain
52	lower surface effective strain
53	lower surface max. principal strain
54	through thickness strain
55	lower surface min. principal strain
56	lower surface effective strain
57	upper surface max. principal strain
58	through thickness strain
59	upper surface min. principal strain
60	upper surface effective strain

Table 8.4. Beam Element Quantities

Component Number	Quantity
1	x-force resultant
2	y-force resultant
3	z-force resultant
4	x-moment resultant
5	y-moment resultant
6	z-moment resultant

For the BINARY option the following cards apply:

Card Format

Card 1	1	2	3	4	5	6	7	8
Variable	NEIPH	NEIPS	MAXINT	STRFLG	SIGFLG	EPSFLG	RLTFLG	ENGFLG
Type	I	I	I	I	I	I	I	I
Default	0	0	3	0	1	1	1	1

Card 2	1	2	3	4	5	6	7	8
Variable	CMPFLG	IEVERP	BEAMIP	DCOMP	SHGE	STSSZ		
Type	I	I	I	I	I	I		
Default	0	0	0	0	0	0		

VARIABLE	DESCRIPTION
NEIPH	Number of additional integration point history variables written to the LS-TAURUS database for solid elements. The integration point data is written in the same order that it is stored in memory-each material model has its own history variables that are stored. For user defined materials it is important to store the history data that is needed for plotting before the data which is not of interest.
NEIPS	Number of additional integration point history variables written to the LS-TAURUS database for both shell and thick shell elements for each integration point, see NEIPH above.
MAXINT	Number of shell integration points written to the LS-TAURUS database, see also *INTEGRATION_SHELL. If the default value of 3 is used then results are output for the outrtmost (top) and innermost (bottom) integration points together with results for the neutral axis. If MAXINT is set to 3 and the the element has 1 integration point then all three results will be the same. If a value other than 3 is used then results for the first MAXINT integration points in the element will be output. Note: If the element has an even number of integration points and MAXINT is not set to 3 then you will not get mid-surface results.
STRFLG	Set to 1 to dump strain tensors for solid, shell and thick shell elements for plotting by LS-TAURUS and ASCII file ELOUT. For shell and thick shell elements two tensors are written, one at the innermost and one at the outermost integration point. For solid elements a singe strain tensor is written.
SIGFLG	Flag for including stress tensor in the shell LS-TAURUS database: EQ.1: include (default), EQ.2: exclude.
SIGFLG	Flag for including stress tensor in the shell LS-TAURUS database: EQ.1: include (default), EQ.2: exclude.
EPSFLG	Flag for including the effective plastic strains in the shell LS-TAURUS database: EQ.1: include (default), EQ.2: exclude.
RLTFLG	Flag for including stress resultants in the shell LS-TAURUS database: EQ.1: include (default), EQ.2: exclude.
ENGFLG	Flag for including internal energy and thickness in the LS-TAURUS database: EQ.1: include (default), EQ.2: exclude.

CMPFLG	Composite material stress output in local coordinate system for shells and solids: EQ.0: global, EQ.1: local.
IEVERP	Every plot state for "d3plot" database is written to a separate file. This option will limit the database to 100 states: EQ.0: more than one state can be on each plotfile, EQ.1: one state only on each plotfile.
BEAMIP	Number of beam integration points for output. This option does not apply to beams that use a resultant formulation.
DCOMP	Data compression to eliminate rigid body data: EQ.1: off (default), no data compression, EQ.2: on.
SHGE	Output shell hourglass energy: EQ.1: off (default), no hourglass energy written, EQ.2: on.
STSSZ	Output shell element time step: EQ.1: off (default), no shell element time step output, EQ.2: on.

*DATABASE_HISTORY_OPTION

Options include:

BEAM
BEAM_SET
NODE
NODE_SET
SHELL
SHELL_SET
SOLID
SOLID_SET
TSHELL
TSHELL_SET

Purpose: Control which nodes or elements are output into the binary history file, D3THDT, the ASCII file NODOUT, and the ASCII file ELOUT. Define as many cards as necessary. The next "*" card terminates the input. See also *DATABASE_BINARY_OPTION and *DATABASE_OPTION.

Card Format

Cards 1,2,... 1 2 3 4 5 6 7 8

Variable	ID1	ID2	ID3	ID4	ID5	ID6	ID7	ID8
Type	I	I	I	I	I	I	I	I

VARIABLE	DESCRIPTION
IDn	NODE/NODE_SET or element/element set ID n. Elements may be BEAM/ BEAM_SET, SHELL/SHELL_SET, SOLID/SOLID_SET, or TSHELL/ TSHELL_SET. The contents of the files are given in Table 8.1 for nodes, Table 8.2 for solid elements, Table 8.3 for shells and thick shells, and Table 8.4 for beam elements. On the binary file D3THDT, the contents may be extended or reduced with the *DATABASE_EXTENT_BINARY definition.

***DATABASE_NODAL_FORCE_GROUP**

Purpose: Define a nodal force group for output into ASCII file NODFOR and the binary file XTFILE. See also *DATABASE_OPTION and *DATABASE_BINARY_OPTION.

Card Format

	1	2	3	4	5	6	7	8
Variable	NSID	CID						
Type	I	I						
Default	none	none						

VARIABLE	DESCRIPTION
NSID	Nodal set ID, see *SET_NODE_OPTION.
CID	Coordinate system ID for output of data in local system, see *DEFINE_ COORDINATE_OPTION.

*DATABASE_SPRING_FORWARD

Purpose: Create spring forward nodal force file. This option is to output resultant nodal force components of sheet metal at the end of the forming simulation into an ASCII file, "SPRING-FORWARD", for spring forward and die corrective simulations.

Card Format

Cards 1	1	2	3	4	5	6	7	8
Variable	IFLAG							
Type	I							

VARIABLE	DESCRIPTION
IFLAG	Output type: EQ.0: off, EQ.1: output element nodal force vector for deformable nodes, EQ.2: output element nodal force vector for materials, subset for NIKE3D interface file.

***DATABASE_SUPERPLASTIC_FORMING**

Purpose: Specify the output intervals to the superplastic forming output files. The option *LOAD_ SUPERPLASTIC_FORMING must be active.

Card Format

Cards 1	1	2	3	4	5	6	7	8
Variable	DTOUT							
Type	F							

VARIABLE	DESCRIPTION
DTOUT	Output time interval for output to “pressure”, “curve1” and “curve2” files. The “pressure” file contains general information from the analysis and the files “curve1” and “curve2” contain pressure versus time from phases 1 and 2 of the analysis. The pressure file may be plotted in Phase 3 of LS-TAURUS using the SUPERPL option.

*DATABASE_TRACER

Purpose: Tracer particles will save a history of either a material point or a spatial point into an ASCII file, TRHIST. This history includes positions, velocities, and stress components. The option *DATABASE_TRHIST must be active.

Card Format

	1	2	3	4	5	6	7	8
Variable	TIME	TRACK	X	Y	Z			
Type	F	I	F	F	F			
Default	0.0	Lagrangian	0	0	0			

VARIABLE	DESCRIPTION
TIME	Start time for tracer particle
TRACK	Tracking option: EQ.0: particle follows material, EQ.1: particle is fixed in space.
X	Initial x-coordinate
Y	Initial y-coordinate
X	Initial z-coordinate

*DEFINE

The keyword *DEFINE provides a way of defining boxes, coordinate systems, load curves, tables, and orientation vectors for various uses. The keyword cards in this section are defined in alphabetical order:

*DEFINE_BOX
*DEFINE_COORDINATE_NODES
*DEFINE_COORDINATE_SYSTEM
*DEFINE_COORDINATE_VECTOR
*DEFINE_CURVE
*DEFINE_SD_ORIENTATION
*DEFINE_TABLE
*DEFINE_VECTOR

*DEFINE_BOX

Purpose: Define a specific box-shaped volume. The two corner points of a box are specified in global coordinates. The box volume is then used for various specifications, e.g., velocities, contact, etc.

Card Format

1 4 5 6

Variable	BOXID	XMN	XMX	YMN	YMX	ZMN	ZMX	
Type	I	F	F	F	F	F	F	
Default	0	0.0	0.0	0.0	0.0	0.0	0.0	
Remarks								

VARIABLE	DESCRIPTION
BOXID	Box ID. Define unique numbers.
XMN	Xmin coordinate.
XMX	Xmax coordinate.
YMN	Ymin coordinate.
YMX	Ymax coordinate.
ZMN	Zmin coordinate.
ZMX	Zmax coordinate.

***DEFINE_COORDINATE_NODES**

Purpose: Define a local coordinate system with three node numbers. The local cartesian coordinate system is defined in the following steps. The z-axis is computed from the cross product of x and $\bar{y}$, (see Figure 9.1), $z = x \times \bar{y}$, then the y-axis is computed via $y = z \times x$.

Card Format

	1	2	3	4	5	6	7	8
Variable	CID	N1	N2	N3				
Type	I	I	I	I				
Default	0	0	0	0				

VARIABLE	DESCRIPTION
CID	Coordinate system ID. A unique number has to be defined.
N1	Number of node located at local origin.
N2	Number of node located along local x-axis.
N3	Number of node located in local x-y plane.

Remark:

1. The nodes N1, N2, and N3 must be separated by a reasonable distance and not colinear to avoid numerical inaccuracies.

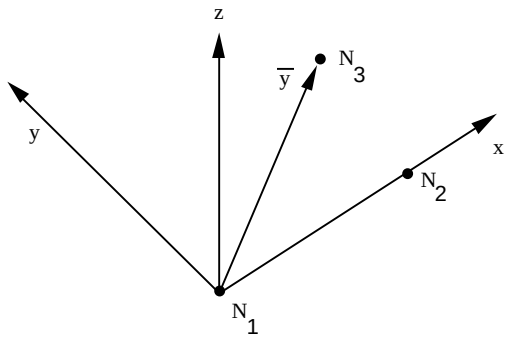


Figure 9.1. Definition of local coordinate system using three nodes.

*DEFINE_COORDINATE_SYSTEM

Purpose: Define a local coordinate system with three points. The same procedure as described in Figure 9.1, see *DEFINE_COORDINATE_NODES, is used. The coordinates of the nodes are given instead. N_1 is defined by (X_0,Y_0,Z_0), N_2 is defined by (X_L,Y_L,Z_L), and N_3 by (X_P,Y_P,Z_P).

Card Format

	1	2	3	4	5	6	7	8
Variable	CID	XO	YO	ZO	XL	YL	ZL	
Type	I	F	F	F	F	F	F	
Default	0	0.0	0.0	0.0	0.0	0.0	0.0	
Remarks								

Second Card

	1	2	3	4	5	6	7	8
Variable	XP	YP	ZP					
Type	F	F	F					
Default	0.0	0.0	0.0					
Remarks								

VARIABLE	DESCRIPTION
CID	Coordinate system ID. A unique number has to be defined.
XO	X-coordinate of origin
YO	Y-coordinate of origin
ZO	Z-coordinate of origin
XL	X-coordinate of point on local x-axis
YL	Y-coordinate of point on local x-axis
ZL	Z-coordinate of point on local x-axis
XP	X-coordinate of point in local x-y plane
YP	Y-coordinate of point in local x-y plane
ZP	Z-coordinate of point in local x-y plane

Remark:

1. The coordinates of the points must be separated by a reasonable distance and not colinear to avoid numerical inaccuracies.

*DEFINE_COORDINATE_VECTOR

Purpose: Define a local coordinate system with two vectors, see Figure 9.2. From the cross product $xy \times x = z$, the z-axis is determined, followed by the computation of the y-axis by $y = z \times x$.

Card Format

	1	2	3	4	5	6	7	8
Variable	CID	XX	YX	ZX	XV	YV	ZV	
Type	I	F	F	F	F	F	F	
Default	0	0.0	0.0	0.0	0.0	0.0	0.0	

VARIABLE	DESCRIPTION
CID	Coordinate system ID. A unique number has to be defined.
XX	X-coordinate on local x-axis. Origin lies at (0,0,0).
YX	Y-coordinate on local x-axis
ZX	Z-coordinate on local x-axis
XV	X-coordinate of local x-y vector
YV	Y-coordinate of local x-y vector
ZV	Z-coordinate of local x-y vector

Remark:

1. These vectors should be separated by a reasonable included angle to avoid numerical inaccuracies.

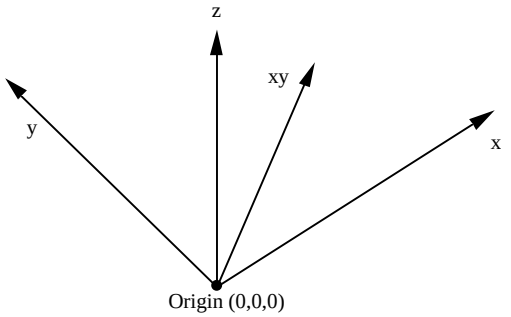


Figure 9.2. Definition of the coordinate system with two vectors.

***DEFINE_CURVE**

Purpose: Define a curve [for example, load (ordinate value) versus time (abcissa value)], often referred to as a load curve.

Card Format

	1	2	3	4	5	6	7	8
Variable	LCID	SIDR	SFA	SFO	OFFA	OFFO	DATTYP	
Type	I	I	F	F	F	F	I	
Default	none	0	1.	1.	0.	0.	0	

Card 2, 3, 4, etc. Put one pair of points per card (2E20.0) Input is terminated when a "*" card is found.

	1	2	3	4	5	6	7	8
Variable	A1		O1					
Type	F		F					
Default	0.0		0.0					

VARIABLE	DESCRIPTION
LCID	Load curve ID. Tables (see *DEFINE_TABLE) and load curves may not share common ID's. LS-DYNA3D allows load curve ID's and table ID's to be used interchangeably. A unique number has to be defined.
SIDR	Stress initialization by dynamic relaxation: EQ.0: load curve used in transient analysis only or for other applications, EQ.1: load curve used in stress initialization but not transient analysis, EQ.2: load curve applies to both initialization and transient analysis.
SFA	Scale factor for abcissa value. This is useful for simple modifications.

VARIABLE	DESCRIPTION
SFO	Scale factor for ordinate value (function). This is useful for simple modifications.
OFFA	Offset for abcissa values, see explanation below.
OFFO	Offset for ordinate values (function), see explanation below.
DATTYP	Data type. Set to 1 for general xy data. This affects how offsets are applied.
A1, A2,...	Abcissa values. Only pairs have to be defined, see remarks below.
O1, O2,...	Ordinate (function) values. Only pairs have to be defined, see remarks below.

Warning: In the definition of Load Curves used in the constitutive models, reasonable spacing of the points should always be observed, i.e., never set a single point off to a value approaching infinity. LS-DYNA3D uses internally discretized curves to improve efficiency in the constitutive models. Also, since the constitutive models extrapolate the curves, it is important to ensure that extrapolation does not lead to physically meaningless values, such as a negative flow stress.

The load curve values are scaled after the offsets are applied.

$$\textit{Abcissa value} = SFA \cdot (\textit{Defined value} + OFFA)$$
$$\textit{Ordinate value} = SFO \cdot (\textit{Defined value} + OFFO)$$

Positive offsets for the load curves (DATTYP=0) are intended for time versus function curves since two additional points are generated automatically at time zero and at time .999*OFFO with the function values set to zero. If DATTYP=1, then the offsets do not create these additional points. Negative offsets for the abcissa simply shifts the abcissa values without creating additional points.

Load curves are not extrapolated by LS-DYNA3D for applied loads such as pressures, concentrated forces, displacement boundary condtions, etc. Function values are set to zero if the time, etc., goes off scale. Therefore, extreme care must be observed when defining load curves. In the constitutive models, extrapolation is employed if the values on the abcissa go off scale.

***DEFINE_SD_ORIENTATION**

Purpose: Define orientation vectors for discrete springs and dampers. Three alternative options are possible. With the first two options, IOP=1 or 2, the vector is defined by coordinates and is fixed permanently in space. The third option orients the vector based on the motion of two nodes, so that the direction can change as the line defined by the nodes rotates.

Card Format

	1	2	3	4	5	6	7	8
Variable	VID	IOP	XT	YT	ZT	NID1	NID2	
Type	I	I	F	F	F	I	I	
Default	0	0	0.0	0.0	0.0	0	0	
Remarks	none		IOP=0,1	IOP=0,1	IOP=0,1	IOP=2	IOP=2	

VARIABLE	DESCRIPTION
VID	Orientation vector ID. A unique number must be used.
IOP	Option: EQ.0: deflections/rotations are measured and forces/moments applied along the following orientation vector. EQ.1: deflections/rotations are measured and forces/moments applied along the axis between the two nodes projected onto the plane normal to the following orientation vector. EQ.2: deflections/rotations are measured and forces/moments applied along a vector defined by the following two nodes.
XT	x-value of orientation vector. Define if IOP=0,1.
YT	y-value of orientation vector. Define if IOP=0,1.
ZT	z-value of orientation vector. Define if IOP=0,1.
NID1	Node 1 ID. Define if IOP=2.
NID2	Node 2 ID. Define if IOP=2.

***DEFINE_TABLE**

Purpose: Define a table. This input section is somewhat unique in that another keyword, ***DEFINE_CURVE**, is used as part of the input in this section. A table consists of a ***DEFINE_TABLE** card followed by n lines of input. Each of the n additional lines define a numerical value in ascending order corresponding to a ***DEFINE_CURVE** input which follows the ***DEFINE_TABLE** keyword and the related input. For example, to define strain rate dependency where it is desired to provide a stress versus strain curve for each strain rate, n strain rates would be defined following the ***DEFINE_TABLE** keyword. The curves then follow which make up the table. There are no rules for defining the n curves, i.e., each curve may have a different origin, spacing, and number of points in their definition. (Load curve ID's defined for the table may be referenced elsewhere in the input.) This rather awkward input is done for efficiency reasons related to the desire to avoid indirect addressing in the inner loops used in the constitutive model stress evaluation.

Card Format

	1	2	3	4	5	6	7	8
Variable	TBID							
Type	I							
Default	none							

Card 2, 3, 4, etc. Put one point per card (E20.0). Input is terminated when a Ò*DEFINE_CURVEÓ card is found.

	1	2	3	4	5	6	7	8
Variable	VALUE							
Type	F							
Default	0.0							

Insert one ***DEFINE_CURVE** input section here for each point defined above.

VARIABLE	DESCRIPTION
TBID	Table ID. Tables and Load curves may not share common ID's. LS-DYNA3D allows load curve ID's and table ID's to be used interchangeably.
VALUE	Load curve will be defined corresponding to this value, e.g., this value could be a strain rate, see purpose above.

Remark:

1. If for example, 10 stress-strain curves for 10 different strain rates are given, 10 cards with the ascending values of strain rate then follow the first card. Afterwards, 10 corresponding *DEFINE_CURVE specifications have to follow.

*DEFINE_VECTOR

Purpose: Define a vector with the coordinates of two nodes.

Card Format

	1	2	3	4	5	6	7	8
Variable	VID	XT	YT	ZT	XH	YH	ZH	
Type	I	F	F	F	F	F	F	
Default	0	0.0	0.0	0.0	0.0	0.0	0.0	
Remarks								

VARIABLE	DESCRIPTION
VID	Vector ID
XT	X-coordinate of tail of vector
YT	Y-coordinate of tail of vector
ZT	Z-coordinate of tail of vector
XH	X-coordinate of head of vector
YH	Y-coordinate of head of vector
ZH	Z-coordinate of head of vector

Remark:

1. The coordinates should differ by a certain margin to avoid numerical inaccuracies.

*DEFORMABLE_TO_RIGID

The cards in this section are defined in alphabetical order and are as follows:

*DEFORMABLE_TO_RIGID
*DEFORMABLE_TO_RIGID_AUTOMATIC
*DEFORMABLE_TO_RIGID_INERTIA

If one of these cards is defined, then any deformable part defined in the model may be switched to rigid during the calculation. Parts that are defined as rigid (*MAT_RIGID) in the input are permanently rigid and cannot be changed to deformable.

Deformable parts may be switched to rigid at the start of the calculation by specifying them on the *DEFORMABLE_TO_RIGID card.

Part switching may be specified on a restart (see RESTART section of this manual) or it may be performed automatically by use of the *DEFORMABLE_TO_RIGID_AUTOMATIC cards.

The *DEFORMABLE_TO_RIGID_INERTIA cards allow inertial properties to be defined for deformable parts that are to be swapped to rigid at a later stage.

It is not possible to perform part material switching on a restart if it was not flagged in the initial analysis. The reason for this is that extra memory needs to be set up internally to allow the switching to take place. If part switching is to take place on a restart, but no parts are to be switched at the start of the calculation, no inertia properties for switching and no automatic switching sets are to be defined, then just define one *DEFORMABLE_TO_RIGID card without further input.

*DEFORMABLE_TO_RIGID

*DEFORMABLE_TO_RIGID

Purpose: Define materials to be switched to rigid at the start of the calculation.

Card Format

	1	2	3	4	5	6	7	8
Variable	PID	MRB						
Type	I	I						
Default	none	0						

VARIABLE	DESCRIPTION
PID	Part ID of the part which is switched to a rigid material, also see *PART.
MRB	Part ID of the master rigid body to which the part is merged. If zero, the part becomes either an independent or master rigid body.

***DEFORMABLE_TO_RIGID_AUTOMATIC**

Purpose: Define a set of parts to be switched to rigid or to deformable at some stage in the calculation.

Card Format

Card 1

	1	2	3	4	5	6	7	8
Variable	SWSET	CODE	TIME 1	TIME 2	TIME 3	ENTNO	RELSW	
Type	I	I	F	F	F	I	I	
Default	none	0	0	1.0E20	0.	0.	0	
Remark		1				1,2		

Card 2

	1	2	3	4	5	6	7	8
Variable	NRBF	NCSF	RWF	DTMAX	D2R	R2D		
Type	I	I	I	F	I	I		
Default	0	0	0	0.	0	0		
Remark								

*DEFORMABLE_TO_RIGID

VARIABLE	DESCRIPTION
SWSET	Set number for this automatic switch set. Must be unique.
CODE	Activation switch code. Defines the test to activate the automatic material switch of the part: EQ.0: switch takes place at time 1, EQ.1: switch takes place between time 1 and time 2 if rigid wall force (specified below) is zero, EQ.2: switch takes place between time 1 and time 2 if contact surface force (specified below) is zero, EQ.3: switch takes place between time 1 and time 2 if rigid wall force (specified below) is non-zero, EQ.4: switch takes place between time 1 and time 2 if contact surface force (specified below) is non-zero.
TIME 1	Switch will not take place before this time.
TIME 2	Switch will not take place after this time: EQ.0 Time 2 set to 1.0e20.
TIME 3	Delay period. After this part switch has taken place, another automatic switch will not take place for the duration of the delay period. If set to zero a part switch may take place immediately after this switch.
ENTNO	Rigid wall/contact surface number for switch codes 1, 2, 3, 4.
RELSW	Related switch set. The related switch set is another automatic switch set that must be activated before this part switch can take place: EQ.0: no related switch set.
NCSF	Flag to delete or activate nodal constraint set. If nodal constraint/spotweld definitions are active in the deformable bodies that are switched to rigid, then the definitions should be deleted to avoid instabilities: EQ.0: no change, EQ.1: delete, EQ.2: activate.
RWF	Flag to delete or activate rigid walls: EQ.0: no change, EQ.1: delete, EQ.2: activate.
DTMAX	Maximum permitted time step size after switch.
D2R	Number of deformable parts to be switched to rigid plus number of rigid parts for which new master/slave rigid body combinations will be defined: EQ.0: no parts defined.
R2D	Number of rigid parts to be switched to deformable: EQ.0: no parts defined.

Remark:

1. Only surface to surface and node node to surface contacts can be used to activate an automatic part switch.

2. Contact surface and rigid wall numbers are the order in which they are defined in the deck. The first rigid wall and the first contact surface encountered in the input deck will have an entity number of 1.

*DEFORMABLE_TO_RIGID

Define D2R cards below:

Card Format

	1	2	3	4	5	6	7	8
Variable	PID	MRB						
Type	I	I						
Default	none	0						

VARIABLE	DESCRIPTION
PID	Part ID of the part which is switched to a rigid material.
MRB	Part ID of the master rigid body to which the part is merged. If zero, the part becomes either an independent or master rigid body.

Define R2D cards below:

Card Format

	1	2	3	4	5	6	7	8
Variable	PID							
Type	I							
Default	none							

VARIABLE	DESCRIPTION
PID	Part ID of the part which is switched to a deformable material.

***DEFORMABLE_TO_RIGID_INERTIA**

Purpose: Inertial properties can be defined for the new rigid bodies that are created when the deformable parts are switched. These can only be defined in the initial input if they are needed in a later restart. Unless these properties are defined, LS-DYNA3D will recompute the new rigid body properties from the finite element mesh. The latter requires an accurate mesh description. **When rigid bodies are merged to a master rigid body, the inertial properties defined for the master rigid body apply to all members of the merged set.**

Card Format

Card 1	1	2	3	4	5	6	7	8
Variable	PID							
Type	I							
Default	none							

Card 2	1	2	3	4	5	6	7	8
Variable	XC	YC	ZC	TM				
Type	F	F	F	F				

Card 3	1	2	3	4	5	6	7	8
Variable	IXX	IXY	IXZ	IYY	IYZ	IZZ		
Type	F	F	F	F	F	F		
Default	none	0.0	0.0	none	0.0	none		

*DEFORMABLE_TO_RIGID

VARIABLE	DESCRIPTION
PID	Part ID, see *PART.
XC	x-coordinate of center of mass
YC	y-coordinate of center of mass
ZC	z-coordinate of center of mass
TM	Translational mass
IXX	I_{xx}, xx component of inertia tensor
IXY	I_{xy}
IXZ	I_{xz}
IYY	I_{yy}
IYZ	I_{yz}
IZZ	I_{zz}

*ELEMENT

The element cards in this section are defined in alphabetical order:

*ELEMENT_BEAM_OPTION
*ELEMENT_DISCRETE
*ELEMENT_MASS
*ELEMENT_SEATBELT
*ELEMENT_SEATBELT_ACCELEROMETER
*ELEMENT_SEATBELT_PRETENSIONER
*ELEMENT_SEATBELT_RETRACTOR
*ELEMENT_SEATBELT_SENSOR
*ELEMENT_SEATBELT_SLIPRING
*ELEMENT_SHELL_OPTION
*ELEMENT_SOLID_OPTION
*ELEMENT_TSHELL

The ordering of the element cards in the input file is competely arbitrary. An arbitrary number of element blocks can be defined preceeded by a keyword control card.

***ELEMENT_BEAM_OPTION**

Available options include:

<BLANK>
THICKNESS

Purpose: Define beam and truss elements. For beams, two alternatives are available. Standard is the stress resultant beam (Belytschko beam) using the < BLANK > option. Using the THICKNESS option, an integration through the thickness is performed for the so-called Hughes-Liu beam and the Belytschko-Schwer beams. Also, so-called discrete beams are defined with this option.

Card Format (10I8)

	1	2	3	4	5	6	7	8	9	10
Variable	EID	PID	N_1	N_2	N_3					
Type	I	I	I	I	I					
Default	none	none	none	none	none					
Remarks					1					

Optional Card (Required if THICKNESS is specified after the keyword)

	1	2	3	4	5	6	7	8	9	10
Resultant beam	A		ISS		ITT		IRR		SA	
Integrated beam	THIC1S (s-dir)		THIC2S (s-dir)		THIC1T (t-dir)		THIC2T (t-dir)			
Type	F		F		F		F		F	
Remarks	2		3		3		3		4	

VARIABLE	DESCRIPTION
EID	Element ID. A unique number has to be specified.
PID	Part ID, see *PART.
N1	Nodal point (end) 1.
N2	Nodal point (end) 2.
N3	Nodal point 3.
THIC1S	Beam thickness in s direction at node 1 for integrated beam.
THIC2S	Beam thickness in s direction at node 2 for integrated beam.
THIC1T	Beam thickness in t direction at node 1 for integrated beam.
THIC2T	Beam thickness in t direction at node 2 for integrated beam.
A	Area for resultant beam.
ISS	Inertia about s axis for resultant beam.
ITT	Inertia about t axis for resultant beam.
IRR	Inertia about r axis for resultant beam.
SA	Shear area for resultant beam.

Remarks:

1. A plane through N_1, N_2, and N_3 defines the orientation of the principal r-s plane of the beam, see Figure 11.1.
2. If the second card is not defined for the resultant beam or if the area, A, is not defined the properties are taken from the cross section cards, see *SECTION_BEAM.
3. Do not define for discrete beams (beam type 6), see *SECTION_BEAM.
4. Define for resultant beam elements only, see *SECTION_BEAM.

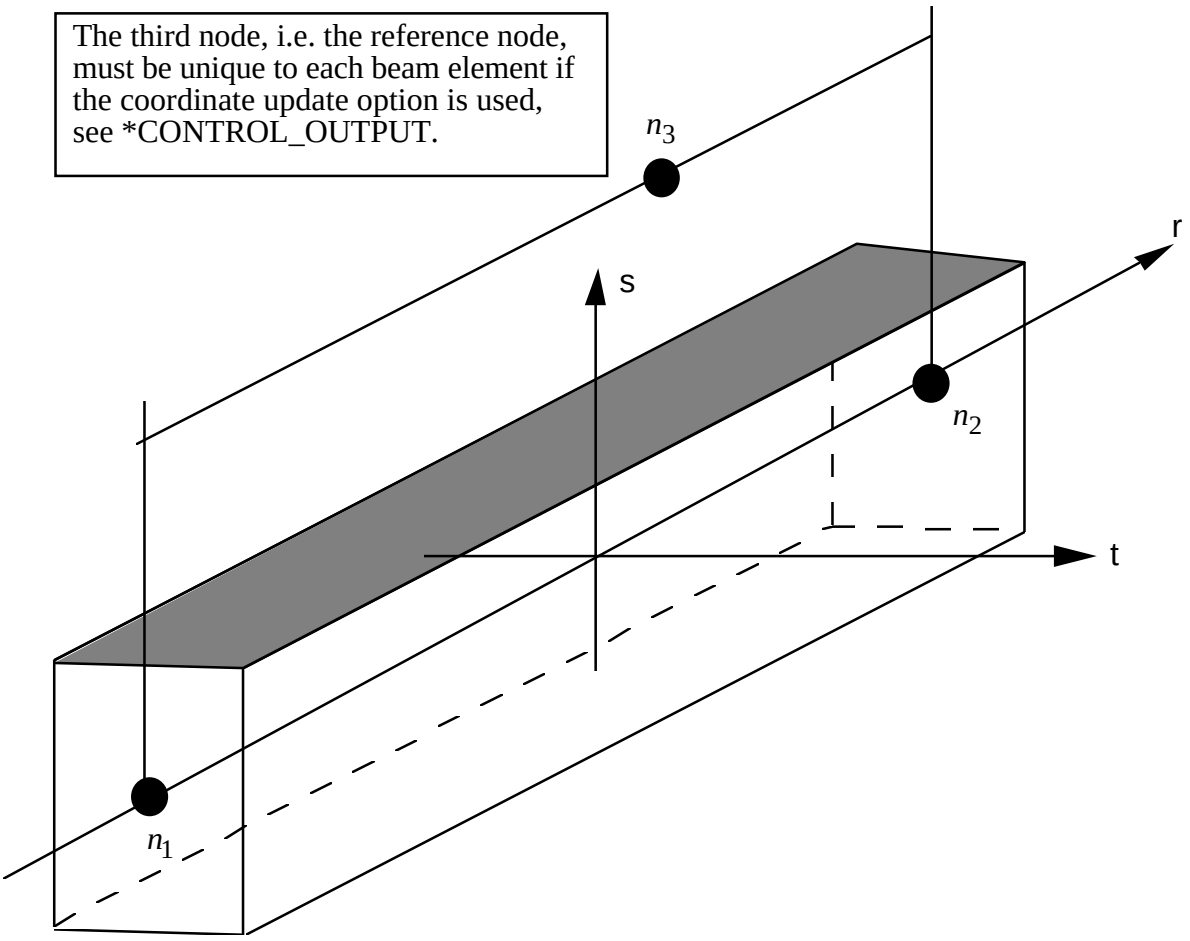

The third node, i.e. the reference node, must be unique to each beam element if the coordinate update option is used, see *CONTROL_OUTPUT.
n3
r
s
n2
t
n1

Figure 11.1. LS-DYNA3D beam elements. Node n_3 determines the initial orientation of the cross section.

***ELEMENT_DISCRETE**

Purpose: Define a discrete (spring or damper) element between two nodes or a node and ground.

Card Format (5I8,E16.0,I8,E16.0)

	1	2	3	4	5	6	7	8	9	10
Variable	EID	PID	N1	N2	VID	S		PF	OFFSET	
Type	I	I	I	I	I	F		I	F	
Default	none	none	none	none	0	1.		0	0	
Remarks									positive for tension	

VARIABLE	DESCRIPTION
EID	Element ID. A unique number has to be used.
PID	Part ID, see *PART.
N1	Nodal point 1.
N2	Nodal point 2. If zero, the spring/damper connects node N1 to ground.
VID	Orientation option: EQ.0: the spring/damper acts along the axis from node N1 to N2, NE.0: the spring/damper acts along the axis defined by the orientation vector, VID defined in the *DEFINE_SD_ORIENTATION section.
S	Scale factor on forces.
PF	Print flag: EQ.0: forces are printed in DEFORC file, see *DATABASE_OPTION, EQ.1: forces are not printed DEFORC file.
OFFSET	Initial offset. The initial offset is a displacement or rotation at time zero. For example, a positive offset on a translational spring will lead to a tensile force being developed at time zero.

*ELEMENT

*ELEMENT_MASS

Purpose: Define a lumped mass element assigned to a nodal point.

Card Format (2I8,E16.0)

	1	2	3	4	5	6	7	8	9	10
Variable	EID	NID	MASS							
Type	I	I	F							
Default	none	none	0.							
Remarks										

VARIABLE	DESCRIPTION
EID	Element ID. A unique number must be used.
NID	Node ID. Node to which the mass is assigned.
MASS	Mass value

***ELEMENT_SEATBELT**

Purpose: Define a seat belt element.

Card Format (5I8,E16.0)

	1	2	3	4	5	6	7	8	9	10
Variable	EID	PID	N_1	N_2	SBRID	SLEN				
Type	I	I	I	I	I	F				
Default	none	none	none	none	none	0.0				
Remarks										

VARIABLE	DESCRIPTION
EID	Element ID. A unique number has to be used.
PID	Part ID
N1	Node 1 ID
N2	Node 2 ID
SBRID	Retractor ID, see *ELEMENT_SEATBELT_RETRACTOR.
SLEN	Initial slack length

Remarks:

1. The retractor ID should be defined only if the element is initially **inside** a retractor, see *ELEMENT_SEATBELT_RETRACTOR.
2. Belt elements are single degree of freedom elements connecting two nodes. When the strain in an element is positive (i.e. the current length is greater then the unstretched length), a tension force is calculated from the material characteristics and is applied along the current axis of the element to oppose further stretching. The unstretched length of the belt is taken as the initial distance between the two nodes defining the position of the element plus the initial slack length.

*ELEMENT_SEATBELT_ACCELEROMETER

Purpose: Define seat belt accelerometer. The accelerometer is fixed to a rigid body containing the three nodes defined below.

Card Format

	1	2	3	4	5	6	7	8
Variable	SBACID	NID1	NID2	NID3				
Type	I	I	I	I				
Default	0	0	0	0				
Remarks								

VARIABLE	DESCRIPTION
SBACID	Accelerometer ID. A unique number has to be used.
NID1	Node 1 ID
NID2	Node 2 ID
NID3	Node 3 ID

The presence of the accelerometer means that the accelerations and velocities of node 1 will be output to **all** output files in local instead of global coordinates.

The local coordinate system is defined by the three nodes as follows:

- local **x** from node 1 to node 2,
- local **z** perpendicular to the plane containing nodes, 1, 2, and 3 (**z** = **x** × **a)**, where **a** is from node 1 to node 3),
- local **y** = **z** × **x.**

The three nodes should all be part of the same rigid body. The local axis then rotates with the body.

***ELEMENT_SEATBELT_PRETENSIONER**

Purpose: Define seat belt pretensioner. A combination with sensors and retractors is also possible.

Card Format

	1	2	3	4	5	6	7	8
Variable	SBPRID	SBPRTY	SBSID1	SBSID2	SBSID3	SBSID4		
Type	I	I	I	I	I	I		
Default	0	0	0	0	0	0		
Remarks	1		2					

Second Card

	1	2	3	4	5	6	7	8
Variable	SBRID	TIME	PTLCID					
Type	I	F	I					
Default	0	0.0	0					
Remarks								

VARIABLE	DESCRIPTION
SBPRID	Pretensioner ID. Use consecutive numbering, see below.
SBPRTY	Pretensioner type: EQ.1: pyrotechnic retractor, EQ.2: pre-loaded spring becomes active, EQ.3: lock spring removed.

VARIABLE	DESCRIPTION
SBSID1	Sensor 1, see *ELEMENT_SEATBELT_SENSOR.
SBSID2	Sensor 2, see *ELEMENT_SEATBELT_SENSOR.
SBSID3	Sensor 3, see *ELEMENT_SEATBELT_SENSOR.
SBSID4	Sensor 4, see *ELEMENT_SEATBELT_SENSOR.
SBRID	Retractor number (SBPRTY = 1) or spring element number (SBPRTY = 2 or 3).
TIME	Time between sensor triggering and pretensioner acting.
PTLCID	Load curve for pretensioner (Time after activation, Pull-in) (SBPRTY = 1).

Remarks:

1. Pretensioner ID's should start at 1 and be consecutive.
2. At least one sensor should be defined.

Pretensioners allow modeling of three types of active devices which tighten the belt during the initial stages of a crash. The first type represents a pyrotechnic device which spins the spool of a retractor, causing the belt to be reeled in. The user defines a pull-in versus time curve which applies once the pretensioner activates. The remaining types represent preloaded springs or torsion bars which move the buckle when released. The pretensioner is associated with any type of spring element including rotational. Note that the preloaded spring, locking spring and any restraints on the motion of the associated nodes are defined in the normal way; the action of the pretensioner is merely to cancel the force in one spring until (or after) it fires. With the second type, the force in the spring element is canceled out until the pretensioner is activated. In this case the spring in question is normally a stiff, linear spring which acts as a locking mechanism, preventing motion of the seat belt buckle relative to the vehicle. A preloaded spring is defined in parallel with the locking spring. This type avoids the problem of the buckle being free to Ôdrift' before the pretensioner is activated.

To activate the pretensioner, the following sequence of events must occur:

1. Any one of up to four sensors must be triggered.
2. Then a user-defined time delay occurs.
3. Then the pretensioner acts.

***ELEMENT_SEATBELT_RETRACTOR**

Purpose: Define seat belt retractor.

Card Format

	1	2	3	4	5	6	7	8
Variable	SBRID	SBRNID	SBID	SID1	SID2	SID3	SID4	
Type	I	I	I	I	I	I	I	
Default	0	0	0	0	0	0	0	
Remarks	1	2		3				

Second Card

	1	2	3	4	5	6	7	8
Variable	TDEL	PULL	LLCID	ULCID	LFED			
Type	F	F	I	I	F			
Default	0.0	0.0	0	0	0.0			
Remarks			4	5				

VARIABLE	DESCRIPTION
SBRID	Retractor ID. Use consecutive numbering, see below.
SBRNID	Retractor node ID
SBID	Seat belt element ID
SID1	Sensor ID 1

VARIABLE	DESCRIPTION
SID2	Sensor ID 2
SID3	Sensor ID 3
SID4	Sensor ID 4
TDEL	Time delay after sensor triggers.
PULL	Amount of pull-out between time delay ending and retractor locking, a length value.
LLCID	Load curve for loading (Pull-out, Force), see Figure 11.3.
ULCID	Load curve for unloading (Pull-out, Force), see Figure 11.3.
LFED	Fed length, see explanation below.

Remarks:

1. Retractor IDs should start at 1 and be consecutive.
2. The retractor node should not be on any belt elements. The element defined should have one node coincident with the retractor node but should not be inside the retractor.
3. At least one sensor should be defined.
4. The first point of the load curve should be (0, T_{min}). T_{min} is the minimum tension. All subsequent tension values should be greater than T_{min}.
5. The unloading curve should start at zero tension and increase monotonically (i.e., no segments of negative or zero slope).

Retractors allow belt material to be paid out into a belt element. Retractors operate in one of two regimes: unlocked when the belt material is paid out, or reeled in under constant tension and locked when a user defined force-pullout relationship applies.

The retractor is initially unlocked, and the following sequence of events must occur for it to become locked:

1. Any one of up to four sensors must be triggered. (The sensors are described below.)
2. Then a user-defined time delay occurs.
3. Then a user-defined length of belt must be paid out (optional).
4. Then the retractor locks.

and once locked, it remains locked.

In the unlocked regime, the retractor attempts to apply a constant tension to the belt. This feature allows an initial tightening of the belt and takes up any slack whenever it occurs. The tension value is taken from the first point on the force-pullout load curve. The maximum rate of pull out or pull in is given by 0.01 × fed length per time step. Because of this, the constant tension value is not always achieved.

In the locked regime, a user-defined curve describes the relationship between the force in the attached element and the amount of belt material paid out. If the tension in the belt subsequently relaxes, a different user-defined curve applies for unloading. The unloading curve is followed until the minimum tension is reached.

The curves are defined in terms of initial length of belt. For example, if a belt is marked at 10mm intervals and then wound onto a retractor, and the force required to make each mark emerge from the (locked) retractor is recorded, the curves used for input would be as follows:

0	Minimum tension (should be > zero)
10mm	Force to emergence of first mark
20mm	Force to emergence of second mark
.	.
.	.
.	.

Pyrotechnic pretensions may be defined which cause the retractor to pull in the belt at a predetermined rate. This overrides the retractor force-pullout relationship from the moment when the pretensioner activates.

If desired, belt elements may be defined which are initially inside the retractor. These will emerge as belt material is paid out, and may return into the retractor if sufficient material is reeled in during unloading.

Elements e2, e3 and e4 are initially inside the retractor, which is paying out material into element e1. When the retractor has fed L_{crit} into e1, where

L_{crit} = fed length - 1.1 × minimum length
(minimum length defined on belt material input)
(fed length defined on retractor input)

element e2 emerges with an unstretched length of 1.1 × minimum length; the unstretched length of element e1 is reduced by the same amount. The force and strain in e1 are unchanged; in e2, they are set equal to those in e1. The retractor now pays out material into e2.

If no elements are inside the retractor, e2 can continue to extend as more material is fed into it.

As the retractor pulls in the belt (for example, during initial tightening), if the unstretched length of the mouth element becomes less than the minimum length, the element is taken into the retractor.

To define a retractor, the user enters the retractor node, the Ômouth' element (into which belt material will be fed), e1 in Figure 11.2, up to 4 sensors which can trigger unlocking, a time delay, a payout delay (optional), load and unload curve numbers, and the fed length. The retractor node is typically part of the vehicle structure; belt elements should not be connected to this node directly, but any other feature can be attached including rigid bodies. The mouth element should have a node coincident with the retractor but should not be inside the retractor. The fed length would typically be set either to a typical element initial length, for the distance between painted marks on a real belt for comparisons with high speed film. The fed length should be at least three times the minimum length.

If there are elements initially inside the retractor (e2, e3 and e4 in the Figure) they should not be referred to on the retractor input, but the retractor should be identified on the element input for these elements. Their nodes should all be coincident with the retractor node and should not be restrained or constrained. Initial slack will automatically be set to 1.1 × minimum length for these elements; this overrides any user-defined value.

Weblockers can be included within the retractor representation simply by entering a Ôlocking up' characteristic in the force pullout curve, see Figure 11.3. The final section can be very steep (but must have a finite slope).

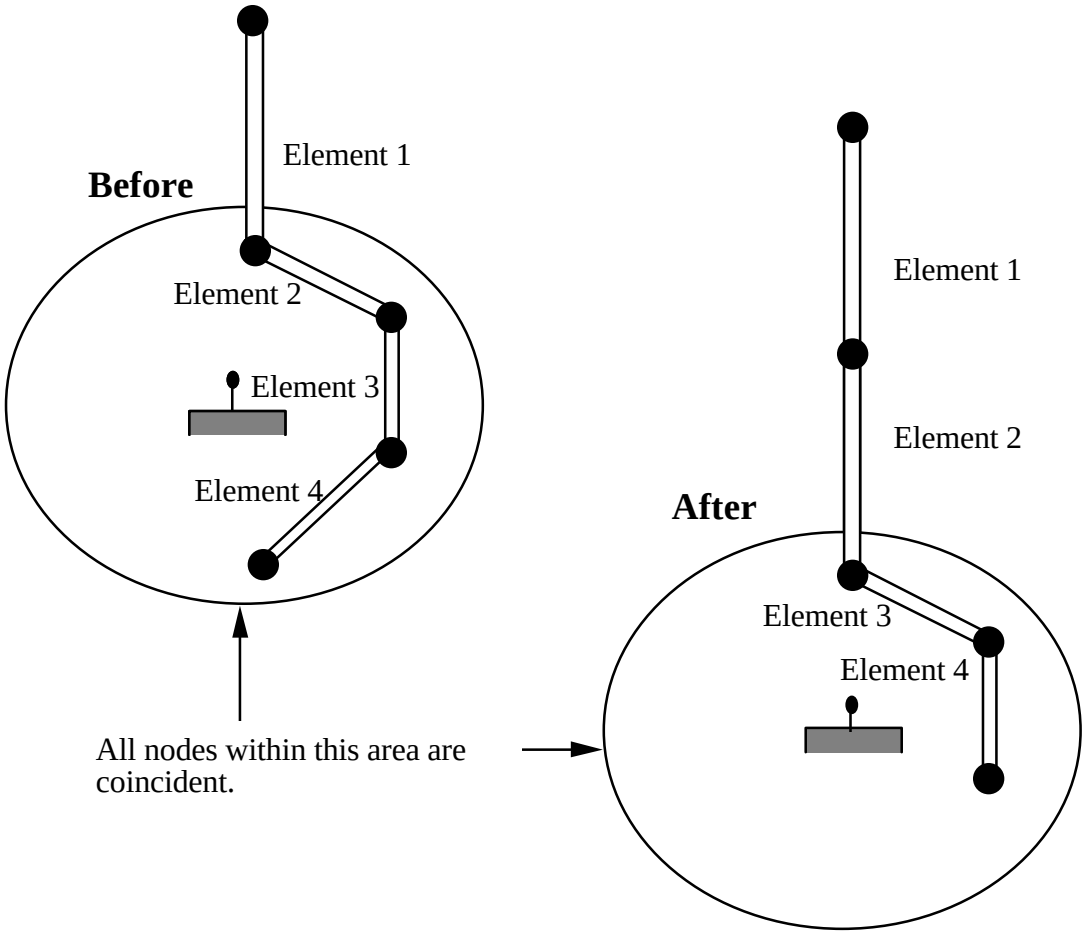


Figure 11.2. Elements in a retractor.

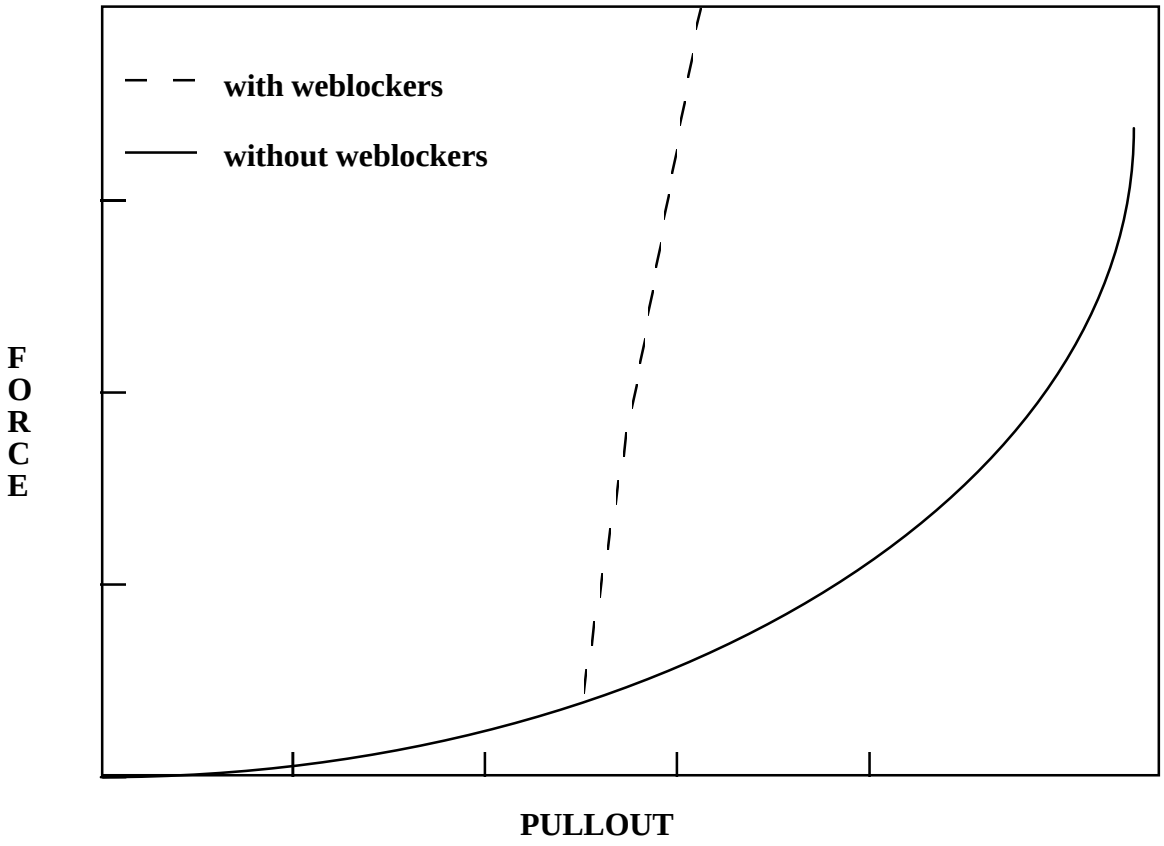


Figure 11.3. Retractor force pull characteristics.

***ELEMENT_SEATBELT_SENSOR**

Purpose: Define seat belt sensor. Four types are possible, see explanation below.

Card Format

	1	2	3	4	5	6	7	8
Variable	SBSID	SBSTYP	SBSFL					
Type	I	I	I					
Default	0	0	0					
Remarks	1							

Second Card if SBSTYP=1

	1	2	3	4	5	6	7	8
Variable	NID	DOF	ACC	ATIME				
Type	I	I	F	F				
Default	0	0	0.0	0.0				
Remarks	2							

Second Card if SBSTYP=2

	1	2	3	4	5	6	7	8
Variable	SBRID	PULRAT	PULTIM					
Type	I	F	F					
Default	0	0.0	0.0					
Remarks								

Second Card if SBSTYP=3

	1	2	3	4	5	6	7	8
Variable	TIME							
Type	F							
Default	0.0							
Remarks								

Second Card if SBSTYP=4

	1	2	3	4	5	6	7	8
Variable	NID1	NID2	DMX	DMN				
Type	I	I	F	F				
Default	0	0	0.0	0.0				
Remarks			3	3				

VARIABLE	DESCRIPTION
SBSID	Sensor ID, see remark below.
SBSTYP	Sensor type: EQ.1: acceleration of node, EQ.2: retractor pull-out rate, EQ.3: time, EQ.4: distance between nodes.
SBSFL	Sensor flag: EQ.0: sensor active during dynamic relaxation, EQ.1: sensor can be triggered during dynamic relaxation.
NID	Node ID of sensor
DOF	Degree of freedom: EQ.1: x, EQ.2: y, EQ.3: z.
ACC	Activating acceleration
ATIME	Time over which acceleration must be exceeded
SBRID	Retractor ID, see *ELEMENT_SEATBELT_RETRACTOR.
PULRAT	Rate of pull-out (length/time units)
PULTIM	Time over which rate of pull-out must be exceeded
TIME	Time at which sensor triggers
NID1	Node 1 ID
NID2	Node 2 ID
DMX	Maximum distance
DMN	Minimum distance

Remarks:

1. Sensor ID's should start at 1 and be consecutive.
2. Node should not be on rigid body, velocity boundary condition, or other Ôimposed motion' feature.
3. Sensor triggers when the distance between the two nodes is $d \geq d_{max}$ or $d \leq d_{min}$.

Sensors are used to trigger locking of retractors and activate pretensioners. Four types of sensors are available which trigger according to the following criteria:

Type 1 – When the magnitude of x-, y-, or z- acceleration of a given node has remained above a given level continuously for a given time, the sensor triggers. This does not work with nodes on rigid bodies.

Type 2 – When the rate of belt payout from a given retractor has remained above a given level continuously for a given time, the sensor triggers.

Type 3 – The sensor triggers at a given time.

Type 4 – The sensor triggers when the distance between two nodes exceeds a given maximum or becomes less than a given minimum. This type of sensor is intended for use with an explicit mass/spring representation of the sensor mechanism.

By default, the sensors are inactive during dynamic relaxation. This allows initial tightening of the belt and positioning of the occupant on the seat without locking the retractor or firing any pretensioners. However, a flag can be set in the sensor input to make the sensors active during the dynamic relaxation phase.

***ELEMENT_SEATBELT_SLIPRING**

Purpose: Define seat belt slip ring.

Card Format

	1	2	3	4	5	6	7	8
Variable	SBSRID	SBID1	SBID2	FC	SBRNID			
Type	I	I	I	F	I			
Default	0	0	0	0.0	0			
Remarks	1				2			

VARIABLE	DESCRIPTION
SBSRID	Slipring ID
SBID1	Seat belt element 1 ID
SBID2	Seat belt element 2 ID
FC	Coulomb friction coefficient
SBRNID	Slip ring node, NID

Remarks:

1. Slip ring ID's should start at 1 and be consecutive.
2. Elements 1 and 2 should share a node which is coincident with the slip ring node. The slip ring node should not be on any belt elements.

Sliprings allow continuous sliding of a belt through a sharp change of angle. Two elements (1 & 2 in Figure 11.4) meet at the slipring. Node B in the belt material remains attached to the slipring node, but belt material (in the form of unstretched length) is passed from element 1 to element 2 to achieve slip. The amount of slip at each timestep is calculated from the ratio of forces in elements 1 and 2. The ratio of forces is determined by the relative angle between elements 1 and 2 and the coefficient of friction, μ. The tension in the belts are taken as T_1 and T_2, where T_2 is

on the high tension side and T_1 is the force on the low tension side. Thus, if T_2 is sufficiently close to T_1, no slip occurs; otherwise, slip is just sufficient to reduce the ratio $T_2 \acute{U} T_1$ to $e^{\mu\Theta}$. No slip occurs if both elements are slack. The out-of-balance force at node B is reacted on the slipring node; the motion of node B follows that of slipring node.

If, due to slip through the slipring, the unstretched length of an element becomes less than the minimum length (as entered on the belt material card), the belt is remeshed locally: the short element passes through the slipring and reappears on the other side (see Figure 11.4). The new unstretched length of e1 is 1.1 × minimum length. Force and strain in e2 and e3 are unchanged; force and strain in e1 are now equal to those in e2. Subsequent slip will pass material from e3 to e1. This process can continue with several elements passing in turn through the slipring.

To define a slipring, the user identifies the two belt elements which meet at the slipring, the friction coefficient, and the slipring node. The two elements must have a common node coincident with the slipring node. No attempt should be made to restrain or constrain the common node for its motion will automatically be constrained to follow the slipring node. Typically, the slipring node is part of the vehicle body structure and, therefore, belt elements should not be connected to this node directly, but any other feature can be attached, including rigid bodies.

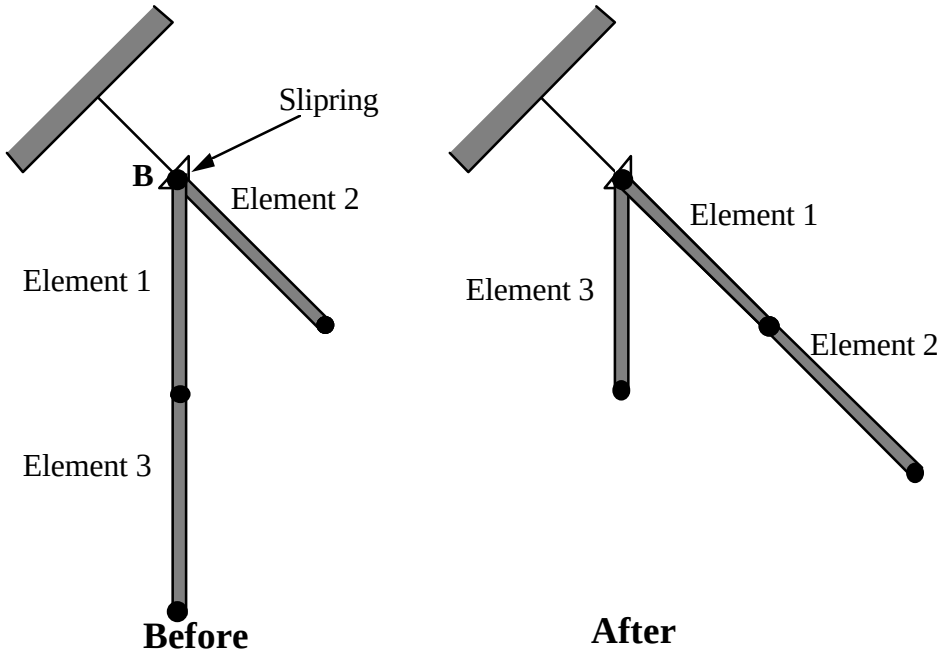


Figure 11.4. Elements passing through slipring.

***ELEMENT_SHELL_OPTION**

Available options include:

<BLANK>
THICKNESS
BETA

Purpose: Define three and four node shell or membrane element. The type of the element is specified through the part ID (see *PART) and the section ID (see *SECTION_SHELL). Also, the thickness of each element can be specified. For orthotropic and anisotropic materials a local material axis can be defined.

Card Format

Card 1	1	2	3	4	5	6	7	8	9	10
Variable	EID	PID	N1	N2	N3	N4				
Type	I	I	I	I	I	I				
Default	none	none	none	none	none	none				
Remarks			3	3	3	3				

Optional Card (Required if THICKNESS or BETA is specified after the keyword)

	1	2	3	4	5	6	7	8	9	10
Variable	THIC1		THIC2		THIC3		THIC4		PSI	
Type	F		F		F		F		F	
Default	0.		0.		0.		0.		0.	
Remarks	1								2	

VARIABLE	DESCRIPTION
EID	Element ID. Unique numbers have to be chosen.
PID	Part ID, see *PART.
N1	Nodal point 1
N2	Nodal point 2
N3	Nodal point 3
N4	Nodal point 4
THIC1	Shell thickness at node 1
THIC2	Shell thickness at node 2
THIC3	Shell thickness at node 3
THIC4	Shell thickness at node 4
PSI	Orthotropic material angle offset measured from the reference (1-2 element side) axis, see remark 4 below.

Remarks:

1. Default values in place of zero shell thicknesses are taken from the cross-section property definition of the PID.

2. Beta is defined only for anisotropic materials.

3. Counterclockwise node numbering determines the top surface, see Figure 11.5.

4. To allow for an arbitrary orientation of the shell elements within the finite element mesh, each ply in the composite has a unique orientation angle which measures the offset from some reference in the element. Each integration point through the shell thickness, typically though not limited to one point per ply, requires the definition of the orientation angle at that point. The reference is determined by the angle ψ, which can be defined for each element on the element card, and is measured from the 1-2 element side. Figures 11.6 and 11.7 depict these angles.

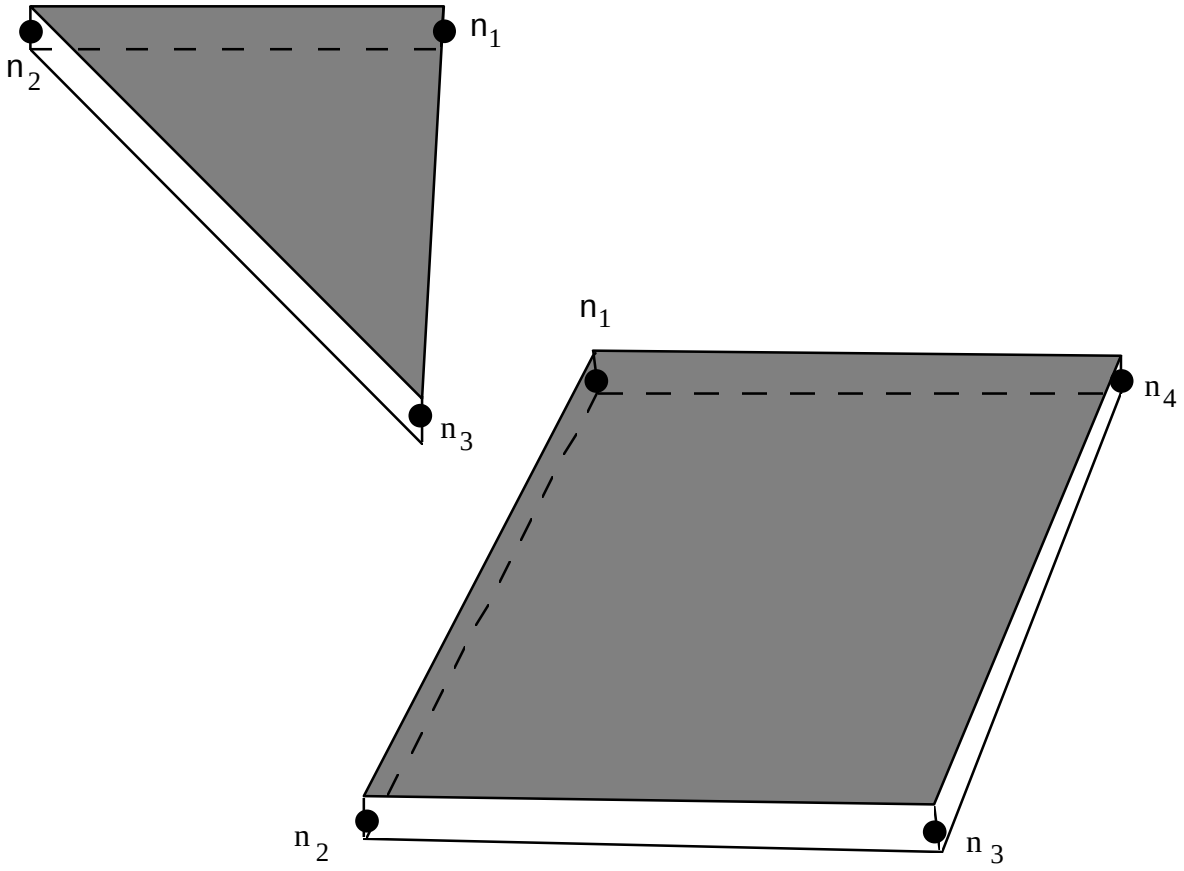


Figure 11.5. LS-DYNA3D shell elements. Counterclockwise node numbering determines the top surface.

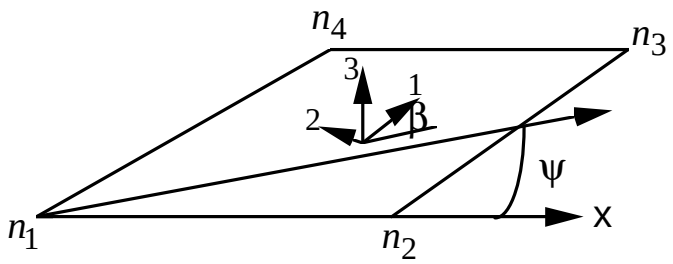


Figure 11.6 Orientation of material directions relative to the 1-2 side.

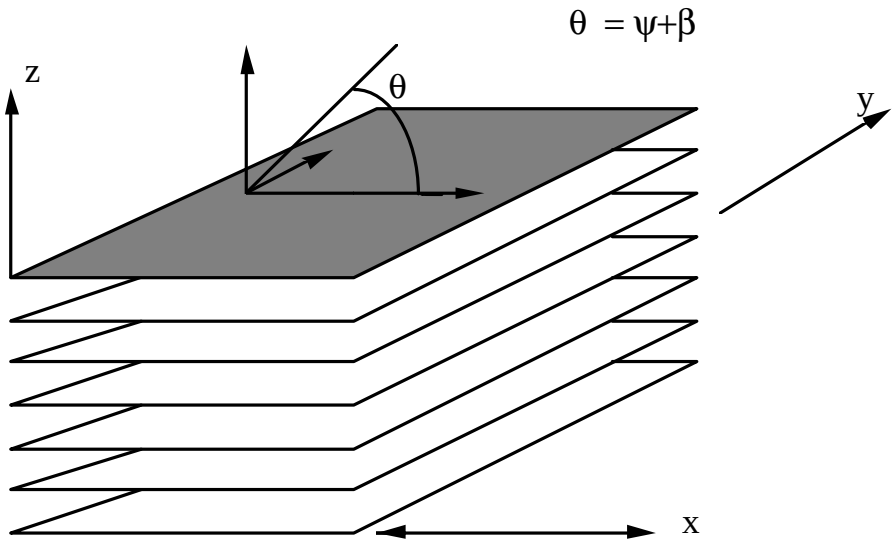


Figure 11.7. A multi-layer laminate can be defined. The angle β_i is defined for the ith lamina (integration point), see *SECTION_SHELL.

*ELEMENT

***ELEMENT_SOLID_OPTION**

Available options include:

<BLANK>
ORTHO

Purpose: Define a solid element. The type of solid element has to be specified via *PART and *SECTION_SOLID_OPTION. Also, a local coordinate system for orthotropic and anisotropic materials can be defined.

Card Format (10I8)

	1	2	3	4	5	6	7	8	9	10
Variable	EID	PID	N1	N2	N3	N4	N5	N6	N7	N8
Type	I	I	I	I	I	I	I	I	I	I
Default	none	none	none	none	none	none	none	none	none	none
Remarks	1									

Optional Cards (Required if ORTHO is specified after the keyword)

Optional card 1	1	2	3	4	5	6	7	8	9	10
Variable	A1		A2		A3					
Type	F		F		F					
Default	0.		0.		0.					
Remarks	2									

Optional card 2	1	2	3	4	5	6	7	8	9	10
Variable	D1		D2		D3					
Type	F		F		F					
Default	0.		0.		0.					
Remarks	2									

VARIABLE	DESCRIPTION
EID	Element ID. A unique number has to be chosen.
PID	Part ID, see *PART.
N1	Nodal point 1
N2	Nodal point 2
N3	Nodal point 3
.	.
.	.
N8	Nodal point 8
A1	x-component of local material direction a.
A2	y-component of local material direction a.
A3	z-component of local material direction a.
D1	x-component of vector in the plane of the material vectors a and b.
D2	y-component of vector in the plane of the material vectors a and b.
D3	z-component of vector in the plane of the material vectors a and b.

Remarks:

1. Four, six, and eight node elements are shown in Figure 11.8. Input of nodes on the element cards for the two degenerate elements is:

   4-node N1, N2, N3, N4, N4, N4, N4, N4

   6-node N1, N2, N3, N4, N5, N5, N6, N6 or N1, N2, N3, N3, N4, N5, N6, N6

2. For the orthotropic and anisotropic material models the local directions may be defined on the second card following the element connectivity definition. The local directions are then computed from the two vectors such that (see Figure 11.9):

$$\underset{\sim}{c} = \underset{\sim}{a} \times \underset{\sim}{d} \text{ and } \underset{\sim}{b} = \underset{\sim}{c} \times \underset{\sim}{a} .$$

   These vectors are internally normalized within LS-DYNA3D.

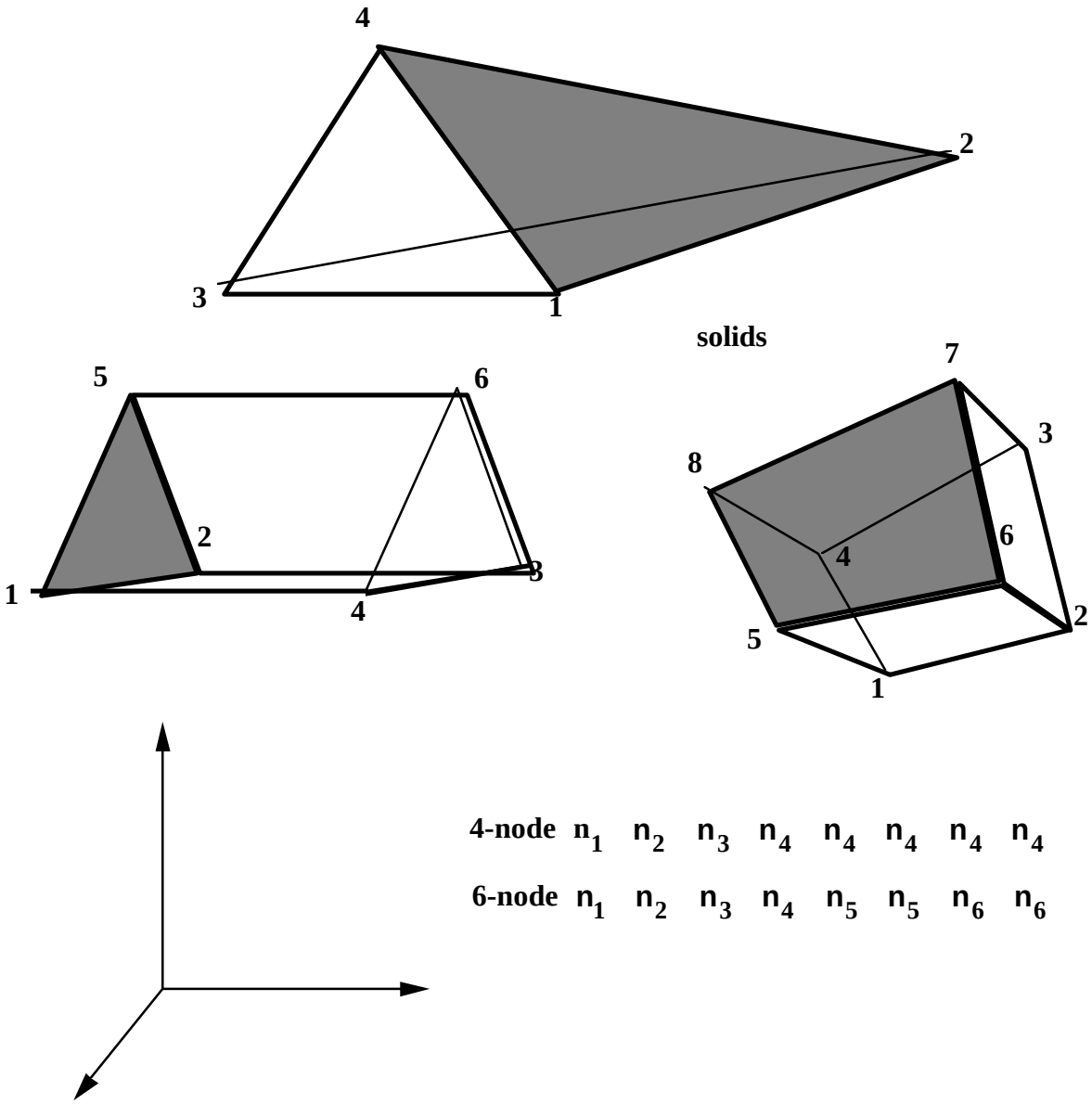


Figure 11.8. Four, six, and eight node solid elements. Nodes 1-4 are on the bottom surface.

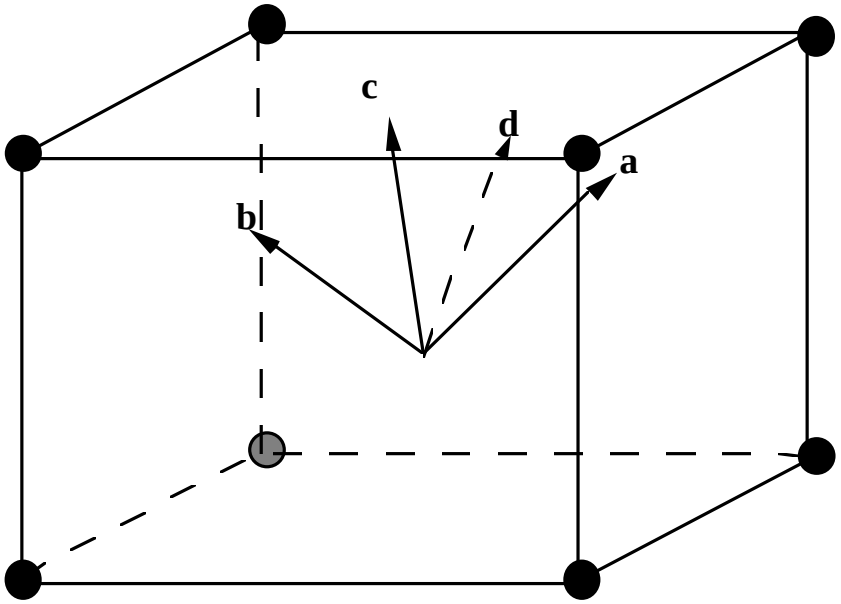


Figure 11.9 Two vectors **a** and **d** are defined and the triad is computed and stored. Vectors **b** and **d** lie in the same plane.

***ELEMENT_TSHELL**

Purpose: Define an eight node thick shell element. This shell element can be used as an alternative to the 4 noded shell elements defined through a midsurface only. The major use is for transition between shell and solid regions or for modelling thick shells. The definition is completed by the *PART and *SECTION_TSHELL cards.

Card Format (10I8)

	1	2	3	4	5	6	7	8	9	10
Variable	EID	PID	N_1	N_2	N_3	N_4	N_5	N_6	N_7	N_8
Type	I	I	I	I	I	I	I	I	I	I
Default	none	none	none	none	none	none	none	none	none	none
Remarks			1							

VARIABLE	DESCRIPTION
EID	Element ID. Unique numbers have to be used.
PID	Part ID, see *PART.
N1	Nodal point 1
N2	Nodal point 2
N3	Nodal point 3
.	.
.	.
N8	Nodal point 8

Remark:

1. The correct numbering of the nodes is essential for correct use. Nodes n_1 to n_4 define the lower surface, and nodes n_5 to n_8 define the upper surface. The integration points lie along the t-axis as depicted in Figure 11.10. Extreme care must be used in defining the connectivity to insure proper orientation.

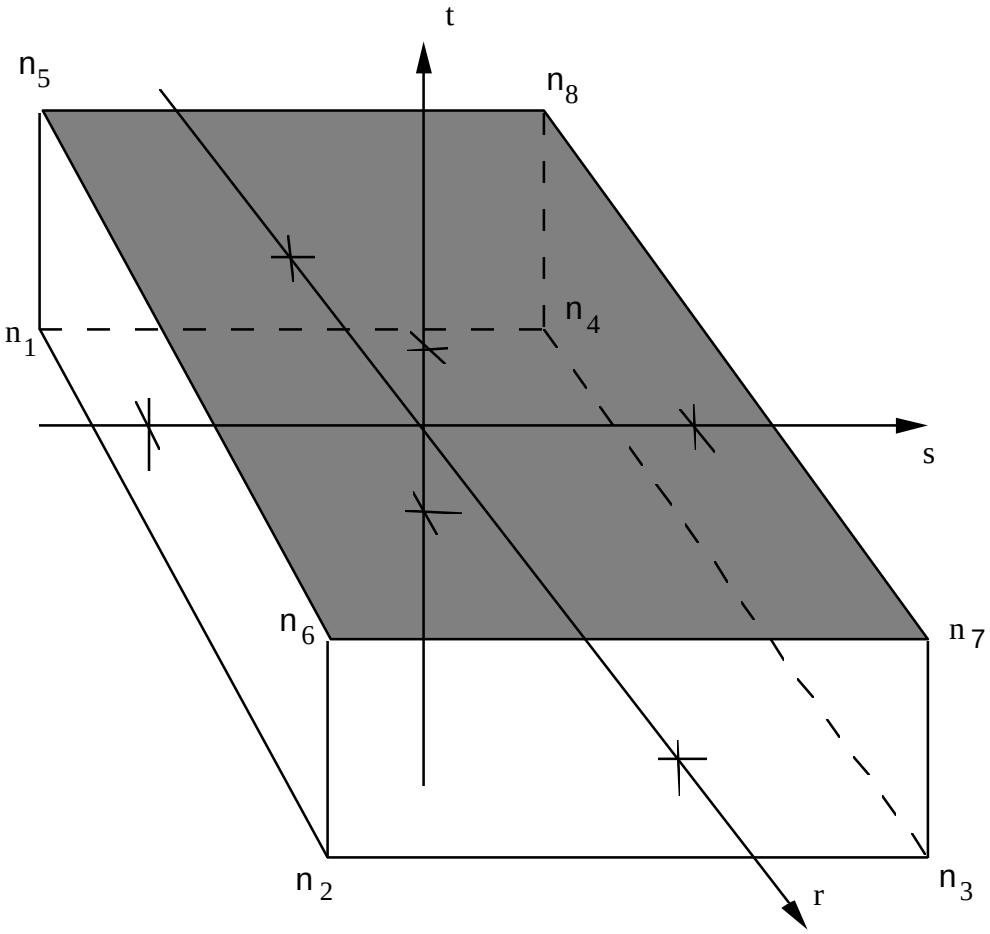


Figure 11.10. Solid 8-node Shell Element.

*EOS

LS-DYNA3D has historically referenced equations of state by type identifiers. Below these identifiers are given with the corresponding keyword name in the order that they appear in the manual. The equations of state can be used with a subset of the materials that are available for solid elements.

TYPE 1:*EOS_LINEAR_POLYNOMIAL
TYPE 2:*EOS_JWL
TYPE 3:*EOS_SACK_TUESDAY
TYPE 4:*EOS_GRUNEISEN
TYPE 5:*EOS_RATIO_OF_POLYNOMIALS
TYPE 6:*EOS_LINEAR_POLYNOMIAL_WITH_ENERGY_LEAK
TYPE 7:*EOS_IGNITION_AND_GROWTH_OF_REACTION_IN_HE
TYPE 8:*EOS_TABULATED_COMPACTION
TYPE 9:*EOS_TABULATED
TYPE 10:*EOS_PROPELLANT_DEFLAGRATION
TYPE 11:*EOS_TENSOR_PORE_COLLAPSE

*EOS_LINEAR_POLYNOMIAL

Purpose: Define coefficients for linear polynomial EOS.

Card Format

Card 1

Variable	EOSID	C0	C1	C2	C3	C 4	C5	C6
Type	I	F	F	F	F	F	F	F

Card 2

Variable	E0	V0						
Type	F	F						

VARIABLE	DESCRIPTION
EOSID	Equation of state label
C0	
C1	
C2	
C3	
C4	
C5	
C6	
E0	Initial internal energy
V0	Initial relative volume

The linear polynomial equation of state is linear in internal energy. The pressure is given by:

$$P = C_0 + C_1\mu + C_2\,\mu^2 + C_3\,\mu^3 + \left(C_4 + C_5\,\mu\ + C_6\,\mu^2\right) E\ .$$

where terms $C_2\mu^2$ and $C_6\mu^2$ are set to zero if $\mu < 0$, $\mu = \frac{\rho}{\rho_0} - 1$, and $\frac{\rho}{\rho_0}$ is the ratio of current density to initial density.

The linear polynomial equation of state may be used to model gas with the gamma law equation of state. This may be achieved by setting:

$$C_0 = C_1 = C_2 = C_3 = C_6 = 0$$

and

$$C_4 = C_5 = \gamma - 1$$

where γ is the ratio of specific heats. The pressure is then given by:

$$p = (\gamma - 1)\frac{\rho}{\rho_0} E$$

The units of E are the units of pressure.

*EOS_JWL

This is Equation of state Form 2.

Card Format

	1	2	3	4	5	6	7	8
Variable	EOSID	A	B	R1	R2	OMEG	E0	VO
Type	I	F	F	F	F	F	F	F

VARIABLE	DESCRIPTION
EOSID	Equation of state label
A	
B	
R1	
R2	
OMEG	
E0	
V0	

The JWL equation of state defines the pressure as

$$p = A\left(1-\frac{\omega}{R_1 V}\right)e^{-R_1 V} + B\left(1-\frac{\omega}{R_2 V}\right)e^{-R_2 V} + \frac{\omega E}{V},$$

and is usually used for detonation products of high explosives.

***EOS_SACK_TUESDAY**

This is Equation of state Form 3.

Card Format

	1	2	3	4	5	6	7	8
Variable	EOSID	A1	A2	A3	B1	B2	E0	V0
Type	I	F	F	F	F	F	F	F

VARIABLE	DESCRIPTION
EOSID	Equation of state label
A1	
A2	
A3	
B1	
B2	
E0	Initial internal energy
V0	Initial relative volume

The Sack equation of state defines pressure as

$$p = \frac{A_3}{V^{A_1}} e^{-A_2 V}\left(1-\frac{B_1}{V}\right)+\frac{B_2}{V}E$$

and is used for detonation products of high explosives.

*EOS_GRUNEISEN

This is Equation of state Form 4.

Card Format

	1	2	3	4	5	6	7	8
Variable	EOSID	C	S1	S2	S3	GAMAO	A	E0
Type	I	F	F	F	F	F	F	F

Card 2

Variable	V0							
Type	F							

VARIABLE	DESCRIPTION
EOSID	Equation of state ID
C	
S1	
S2	
S3	
GAMAO	
A	
E0	Initial internal energy
V0	Initial relative volume

The Gruneisen equation of state with cubic shock velocity-particle velocity defines pressure for compressed materials as

$$p = \frac{\rho_0 C^2 \mu \left[1 + \left(1 - \frac{\gamma_0}{2}\right)\mu - \frac{a}{2}\mu^2\right]}{\left[1 - (S_1 - 1)\mu - S_2 \frac{\mu^2}{\mu+1} S_3 \frac{\mu 3}{\mu+1}\right]^2} + (\gamma_0 + a\mu)E .$$

and for expanded materials as

$$p = \rho_0 C^2 \mu + (\gamma_0 + a\mu)E .$$

where C is the intercept of the v_s-v_p curve; S_1, S_2, and S_3 are the coefficients of the slope of the v_s-v_p curve; γ_0 is the Gruneisen gamma; a is the first order volume correction to $\gamma_{0;}$ and $\mu = \frac{\rho}{\rho_0} - 1.$

*EOS_RATIO_OF_POLYNOMIALS

This is Equation of state Form 5.

Card Format (I10) for card 1, **(4E20.0)** all following cards.

Card 1 1

Variable	EOSID
Type	I

Card 2 1 2 3 4

Variable	A10	A11	A12	A13
Type	F	F	F	F

Card 3 1 2 3 4

Variable	A20	A21	A22	A23
Type	F	F	F	F

Card 4 1 2 3 4

Variable	A30	A31	A32	A33
Type	F	F	F	F

Card 5 1 2 3 4

Variable	A40	A41	A42	A43
Type	F	F	F	F

Card 6	1	2	3	4
Variable	A50	A51	A52	A53
Type	F	F	F	F

Card 7	1	2	3	4
Variable	A60	A61	A62	A63
Type	F	F	F	F

Card 8	1	2	3	4
Variable	A70	A71	A72	A73
Type	F	F	F	F

Card 9	1	2		
Variable	A14	A24		
Type	F	F		

Card 10	1	2	3	4
Variable	ALPH	BETA	E0	V0
Type	F	F	F	F

*EOS

VARIABLE	DESCRIPTION
EOSID	Equation of state ID
A10	
A11	
A12	
A13	
A20	
A21	
A22	
A23	
A30	
A31	
A32	
A33	
A40	
A41	
A42	
A43	
A50	
A51	
A52	
A53	
A60	
A61	
A62	
A63	
A70	
A71	

VARIABLE	DESCRIPTION
A72	
A73	
A14	
A24	
ALPHA	α
BETA	β
E0	Initial internal energy
V0	Initial relative volume

The ratio of polynomials equation of state defines the pressure as

$$p = \frac{F_1 + F_2 E + F_3 E^2 + F_4 E^3}{F_5 + F_6 E + F_7 E^2}(1 + \alpha\mu)$$

where

$$F_i = \sum_{j=0}^{n} A_{ij}\,\mu^j \quad n = 4 \text{ if } i < 3$$

$$\mu = \frac{\rho}{\rho_0} - 1 \qquad n = 3 \text{ if } i \geq 3$$

In expanded elements F_1 is replaced by $F'_1 = F_1 + \beta\,\mu^2$. By setting coefficient $A_{10} = 1.0$, the delta-phase pressure modeling for this material will be initiated. The code will reset it to 0.0 after setting flags.

*EOS_LINEAR_POLYNOMIAL_WITH_ENERGY_LEAK

Card Format

Card 1

Variable	EOSID	C0	C1	C2	C3	C4	C5	C6
Type	I	F	F	F	F	F	F	F

Card 2

Variable	E0	V0	LCID					
Type	F	F	I					

VARIABLE	DESCRIPTION
EOSID	Equation of state label
C0	
C1	
C2	
C3	
C4	
C5	
C6	
E0	Initial internal energy
V0	Initial relative volume
LCID	Load curve ID defining the energy deposition rate

*EOS_IGNITION_AND_GROWTH_OF_REACTION_IN_HE

Card Format

Card 1

Variable	EOSID	AP	BP	R1P	R2P	OMCP	I	G
Type	F	F	F	F	F	F	F	F

Card 2

Variable	H	AE	BE	R1E	R2E	OMCE	FCRIT	Z
Type	F	F	F	F	F	F	F	F

Card 3

Variable	X	Y	CP	CE	M	E0	T0	
Type	F	F	F	F	F	F	F	

VARIABLE	DESCRIPTION
EOSID	Equation of state label
AP	
BP	
R1P	
R2P	
OMCP	
I	First ignition coefficient

VARIABLE	DESCRIPTION
G	Second ignition coefficient
H	Growth coefficient
AE	
BE	
R1E	
R2E	
OMCE	ω
FCRIT	Critical fraction reacted (usually = 1.0)
Z	Pressure exponent
X	
Y	
CP	Heat capacity of reaction products
CE	Heat capacity of unreacted HE
M	Generally = 0
E0	Initial energy of HE per unit volume
T0	Initial temperature (¡K)

A JWL equation of state defines the pressure in the unreacted HE as

$$P_e = A_e\left(1-\frac{\omega_e}{R1_eV_e}\right)e^{-R1_eV_e} + B_e\left(1-\frac{\omega_e}{R2_eV_e}\right)e^{-R2_eV_e} + \frac{\omega E_e}{V_e}$$

where V_e is the relative volume, E_e is the internal energy, and the constants A_e, B_e, ω_e, $R1_e$, and $R2_e$ are inputs.

Similarly, the pressure in the reaction products is defined by another JWL form

$$P_p = A_p\left(1-\frac{\omega_p}{R1_pV_p}\right)e^{-R1_pV_p} + B_e\left(1-\frac{\omega_p}{R2_pV_p}\right)e^{-R2_pV_p} + \frac{\omega E_p}{V_p}$$

The mixture of unreacted explosive and reaction products is defined by the fraction reacted FÊ(FÊ=Ê0Ê→ no reaction, FÊ=Ê1Ê→ complete conversion from explosive to products). The pressures and temperature are assumed to be in equilibrium and the volumes are assumed to be additive.

$$V = (1 - F)\, V_e + F V_p$$

The rate of reaction is

$$\frac{\partial F}{\partial t} = I\,(\mathrm{FCRIT} - F)^y \left(V_e^{-1} - 1\right)^3 \left[1 + G\left(V_e^{-1} - 1\right)\right] + H\,(1 - F)^y F^x P^z \left(V_p^{-1} - 1\right) m$$

where I, G, H, x, y, z, and m (generally m=0) are input constants.

The JWL equations of state and the reaction rates have been fitted to one- and two-dimensional shock initiation and detonation data for four explosives: PBX-9404, RX-03-BB, PETN, and cast TNT. The details of the calculational method are described by Cochran and Chan [1979]. The detailed one-dimensional calculations and parameters for the four explosives are given by Lee and Tarver [1980].

*EOS_TABULATED_COMPACTION

Card Format

Card 1	1	2	3	4	5	6	7	8
Variable	EOSID	GAMA	E0	V0				
Type	I	F	F	F				

Card Format (5E16.0)

Card 2	1	2	3	4	5
Variable	EV1	EV2	EV3	EV4	EV5
Type	F	F	F	F	F

Card 3

Variable	EV6	EV7	EV8	EV9	EV10
Type	F	F	F	F	F

Repeat Cards 2 and 3 for C_i, T_i, and K_i . A total of 9 cards must be defined.

VARIABLE	DESCRIPTION
EOSID	Equation of state label
εV1,εV2,..εVN	ln V
C1,C2,..CN	
T1,T2,..TN	
K1,K2,..KN	

VARIABLE	DESCRIPTION
GAMA	γ
E0	Initial internal energy
V0	Initial relative volume

The tabulated compaction model is linear in internal energy. Pressure is defined by

$$p = C(\varepsilon_V) = gT(\varepsilon_V)\, E$$

in the loading phase. The volumetric strain, ε_V is given by the natural logarithm of the relative volume. Unloading occurs along the unloading bulk modulus to the pressure cutoff. Reloading always follows the unloading path to the point where unloading began, and continues on the loading path, see Figure 12.1. Up to 10 points and as few as 2 may be used when defining the tabulated functions. LS-DYNA3D will extrapolate to find the pressure if necessary.

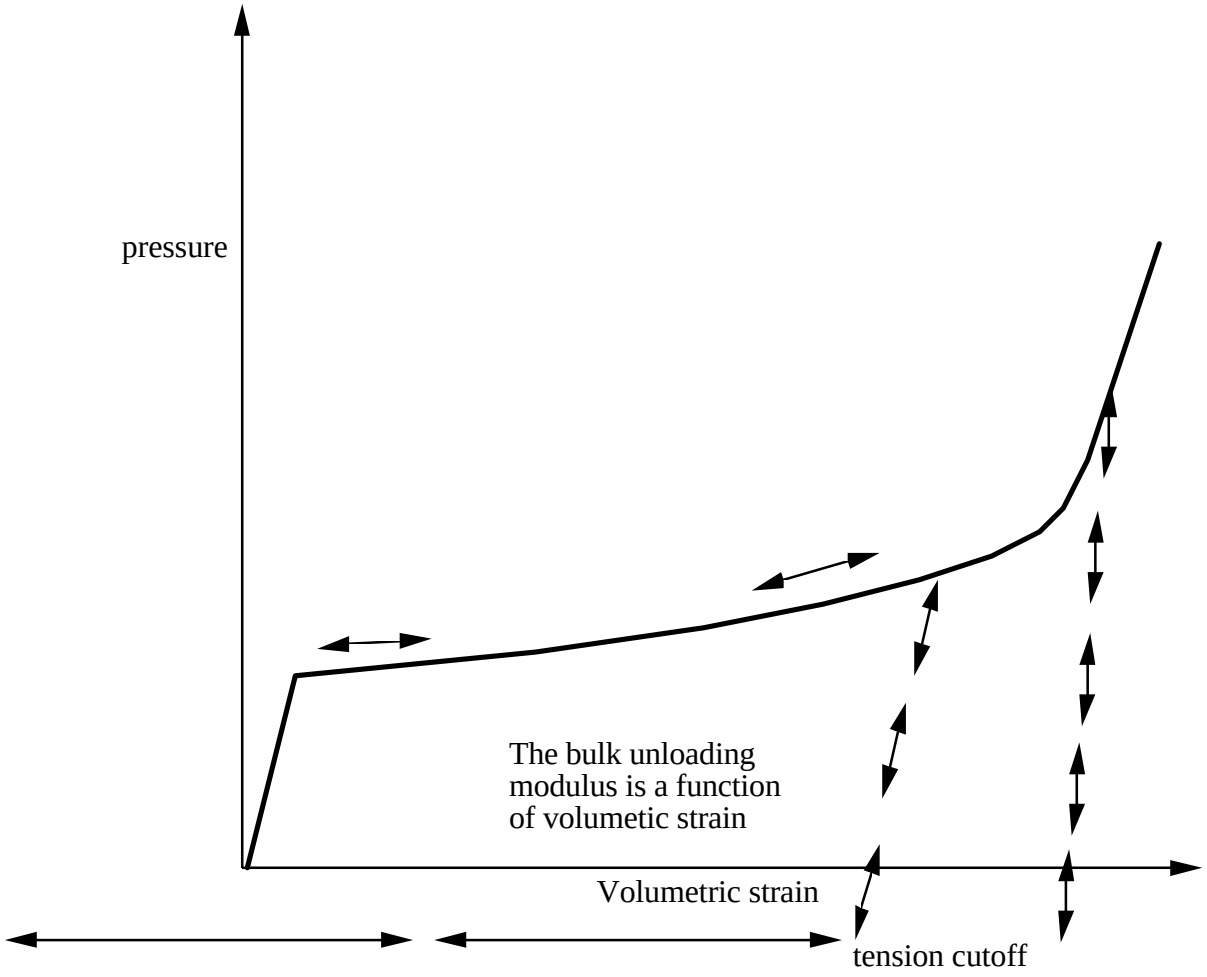


Figure 12.1. Pressure versus volumetric strain curve for Equation of state Form 8 with compaction. In the compacted states the bulk unloading modulus depends on the peak volumetric strain.

***EOS_TABULATED**

This is Equation of state Form 9.

Card Format

Card 1	1	2	3	4	5	6	7	8
Variable	EOSID	GAMA	E0	VO				
Type	I	F	F	F				

Card Format (5E16.0)

Card 2	1	2	3	4	5
Variable	EV1	EV2	EV3	EV4	EV5
Type	F	F	F	F	F

Card 3

Variable	EV6	EV7	EV8	EV9	EV10
Type	F	F	F	F	F

Repeat Cards 2 and 3 for C_i and T_i. A total of 7 cards must be defined.

VARIABLE	DESCRIPTION
EOSID	Equation of state label
εV1,εV2,..εVN	ln V
C1,C2,..CN	
T1,T2,..TN	

VARIABLE	DESCRIPTION
GAMA	γ
E0	Initial internal energy
V0	Initial relative volume

The tabulated equation of state model is linear in internal energy. Pressure is defined by

$$P = C(\varepsilon_V) + \gamma T(\varepsilon_V) E$$

The volumetric strain, ε_V is given by the natural logarithm of the relative volume. Up to 10 points and as few as 2 may be used when defining the tabulated functions. LS-DYNA3D will extrapolate to find the pressure if necessary.

***EOS_PROPELLANT_DEFLAGRATION**

This Equation of state (10) has been added to model airbag propellants.

Card Format

Card 1

Variable	EOSID	A	B	XP1	XP2	FRER		
Type	F	F	F	F	F	F		

Card 2

Variable	R1	R2	R3	R4	R5			
Type	F	F	F	F	F			

Card 3

Variable	R6	FMXIG	FREQ	GROW1	EM			
Type	F	F	F	F	F			

Card 4

Variable	AR1	ES1	CVP	CVR	EETAL	CCRIT	ENQ	TMP0
Type	F	F	F	F	F			

Card 5

Variable	GROW2	AR2	ES2	EN	FMXGR	FMNGR		
Type	F	F	F	F	F	F		

VARIABLE	DESCRIPTION
A	Product JWL coefficient
B	Product JWL coefficient
XP1	Product JWL coefficient
XP2	Product JWL coefficient
FRER	Unreacted Co-volume
G	Product wC_V
R1	Unreacted JWL coefficient
R2	Unreacted JWL coefficient
R3	Unreacted wC_V
R5	Unreacted JWL coefficient
R6	Unreacted JWL coefficient
FMXIG	Initial Fraction Reacted F_o
FREQ	Initial Pressure P_o
GROW1	First burn rate coefficient
EM	Pressure Exponent (1^{st} term)
AR1	Exponent on F (1^{st} term)
ES1	Exponent on (1-F) (1^{st} term)
CVP	Heat capacity products
CVR	Heat capacity unreacted
EETAL	Extra, not presently used
CCRIT	Product co-volume
ENQ	Heat of Reaction
TMP0	Initial Temperature (298¡K)
GROW2	Second burn rate coefficient

VARIABLE	DESCRIPTION
AR2	Exponent on F (2nd term)
ES2	Exponent on (1-F) (2nd term)
EN	Pressure Exponent (2nd term)
FMXGR	Maximum F for 1st term
FMNGR	Minimum F for 2nd term

A deflagration (burn rate) reactive flow model requires an unreacted solid equation of state, a reaction product equation of state, a reaction rate law and a mixture rule for the two (or more) species. The mixture rule for the standard ignition and growth model [Lee and Tarver 1980] assumes that both pressures and temperatures are completely equilibrated as the reaction proceeds. However, the mixture rule can be modified to allow no thermal conduction or partial heating of the solid by the reaction product gases. For this relatively slow process of airbag propellant burn, the thermal and pressure equilibrium assumptions are valid. The equations of state currently used in the burn model are the JWL, Gruneisen, the van der Waals co-volume, and the perfect gas law, but other equations of state can be easily implemented. In this propellant burn, the gaseous nitrogen produced by the burning sodium azide obeys the perfect gas law as it fills the airbag but may have to be modelled as a van der WaalÕs gas at the high pressures and temperatures produced in the propellant chamber. The chemical reaction rate law is pressure, particle geometry and surface area dependant, as are most high pressure burn processes. When the temperature profile of the reacting system is well known, temperature dependent Arrhenius chemical kinetics can be used.

Since the airbag propellant composition and performance data are company private information, it is very difficult to obtain the required information for burn rate modeling. However, Imperial Chemical Industries (ICI) Corporation supplied pressure exponent, particle geometry, packing density, heat of reaction, and atmospheric pressure burn rate data which allowed us to develop the numerical model presented here for their NaN_3 + Fe_2O_3 driver airbag propellant. The deflagration model, its implementation, and the results for the ICI propellant are presented in [Hallquist, et.al., 1990].

The unreacted propellant and the reaction product equations of state are both of the form:

$$p = Ae^{-R_1V} + Be^{-R_2V} + \frac{\omega C_V T}{V - d}$$

where p is pressure (in Mbars), V is the relative specific volume (inverse of relative density), ω is the Gruneisen coefficient, C_v is heat capacity (in Mbars -cc/cc¡K), T is temperature in ¡K, d is the co-volume, and A, B, R_1 and R_2 are constants. Setting A=B=0. yields the van der WaalÕs co-volume equation of state. The JWL equation of state is generally useful at pressures above several kilobars, while the van der WaalÕs is useful at pressures below that range and above the range for which the perfect gas law holds. Of course, setting A=B=d=0 yields the perfect gas law. If accurate values of ω and C_v plus the correct distribution between ÒcoldÓ compression and internal energies are used, the calculated temperatures are very reasonable and thus can be used to check propellant performance.

The reaction rate used for the propellant deflagration process is of the form:

$$\frac{\partial F}{\partial t} = \underbrace{Z(1-F)^y F^x p^w}_{\text{for } 0<F<F_{limit1}} + \underbrace{V(1-F)^u Frp^s}_{\text{for } F_{limit2}<F<1}$$

where F is the fraction reacted (F = 0 implies no reaction, F = 1 is complete reaction), t is time, and p is pressure (in Mbars), r,s,u,w,x,y, F_{limit1} and F_{limit2} are constants used to describe the pressure dependance and surface area dependence of the reaction rates. Two (or more) pressure dependant reaction rates are included in case the propellant is a mixture or exhibited a sharp change in reaction rate at some pressure or temperature. Burning surface area dependences can be approximated using the $(1\text{-F})^y$ F^x terms. Other forms of the reaction rate law, such as Arrhenius temperature dependent $e^{-E/RT}$ type rates, can be used, but these require very accurate temperatures calculations. Although the theoretical justification of pressure dependent burn rates at kilobar type pressures is not complete, a vast amount of experimental burn rate versus pressure data does demonstrate this effect and hydrodynamic calculations using pressure dependent burn accurately simulate such experiments.

The deflagration reactive flow model is activated by any pressure or particle velocity increase on one or more zone boundaries in the reactive material. Such an increase creates pressure in those zones and the decomposition begins. If the pressure is relieved, the reaction rate decreases and can go to zero. This feature is important for short duration, partial decomposition reactions. If the pressure is maintained, the fraction reacted eventually reaches one and the material is completely converted to product molecules. The deflagration front rates of advance through the propellant calculated by this model for several propellants are quite close to the experimentally observed burn rate versus pressure curves.

To obtain good agreement with experimental deflagration data, the model requires an accurate description of the unreacted propellant equation of state, either an analytical fit to experimental compression data or an estimated fit based on previous experience with similar materials. This is also true for the reaction products equation of state. The more experimental burn

rate, pressure production and energy delivery data available, the better the form and constants in the reaction rate equation can be determined.

Therefore the equations used in the burn subroutine for the pressure in the unreacted propellant

$$P_u = R1 \cdot e^{-R5 \cdot V_u} + R2 \cdot e^{-R6 \cdot V_u} + \frac{R3 \cdot T_u}{V_u - FRER}$$

where V_u and T_u are the relative volume and temperature respectively of the unreacted propellant. The relative density is obviously the inverse of the relative volume. The pressure *Pp* in the reaction products is given by:

$$P_p = A \cdot e^{-XP1 \cdot V_p} + B \cdot e^{-XP2 \cdot V_p} + \frac{G \cdot Tp}{V_p - CCRIT}$$

As the reaction proceeds, the unreacted and product pressures and temperatures are assumed to be equilibrated ($T_u = T_p = T$, $p = P_u = P_p$) and the relative volumes are additive:

$$V = (1 - F) \cdot V_u + F \cdot V_p$$

where *V* is the total relative volume. Other mixture assumptions can and have been used in different versions of DYNA2D/3D. The reaction rate law has the form:

$$\frac{\partial F}{\partial t} = \text{GROW1}(\text{P} + \text{FREQ})^{\text{EM}} (\text{F} + \text{FMXIG})^{\text{AR1}} (1 - \text{F} + \text{FMXIG})^{\text{ES1}} + \text{GROW2}(\text{P} + \text{FREQ})^{\text{EN}} (\text{F} + \text{FMXIG})^{\text{AR2}} (1 - \text{F} + \text{FMXIG})^{\text{ES2}}$$

If *F* exceeds FMXGR, the GROW1 term is set equal to zero, and, if *F* is less than FMNGR, the GROW2 term is zero. Thus, two separate (or overlapping) burn rates can be used to describe the rate at which the propellant decomposes.

This equation of state subroutine is used together with a material model to describe the propellant. In the airbag propellant case, a null material model (type #10) can be used. Material type #10 is usually used for a solid propellant or explosive when the shear modulus and yield strength are defined. The propellant material is defined by the material model and the unreacted equation of state until the reaction begins. The calculated mixture states are used until the reaction is complete and then the reaction product equation of state is used. The heat of reaction, ENQ, is assumed to be a constant and the same at all values of F but more complex energy release laws could be implemented.

*EOS_TENSOR_PORE_COLLAPSE

This is Equation of state Form 11.

Card Format

Card 1	1	2	3	4	5	6	7	8
Variable	EOSID	NLD	NCR	MU1	MU2	IE0	EC0	
Type	I	F	F	F	F	F	F	

Repeat Cards 2, etc. as required for ECC_i and PC_i.

VARIABLE	DESCRIPTION
EOSID	Equation of state label
NLD	Virgin loading load curve ID
NCR	Completely crushed load curve ID
MU1	Excess Compression required before any pores can collapse
MU2	Excess Compression point where the Virgin Loading Curve and the Completely Crushed Curve intersect
IE0	Initial Internal Energy
EC0	Initial Excess Compression

The pore collapse model described in the TENSOR manual [23] is no longer valid and has been replaced by a much simpler method. This is due in part to the lack of experimental data required for the more complex model. It is desired to have a close approximation of the TENSOR model in the DYNA code to enable a quality link between them. The TENSOR model defines two curves, the virgin loading curve and the completely crushed curve as shown in Figure 12.2. It also defines the excess compression point required for pore collapse to begin (μ_1), and the excess compression point required to completely crush the material (μ_2). From this data and the maximum excess compression the material has attained (μ_{max}), the pressure for any excess compression (μ) can be determined.

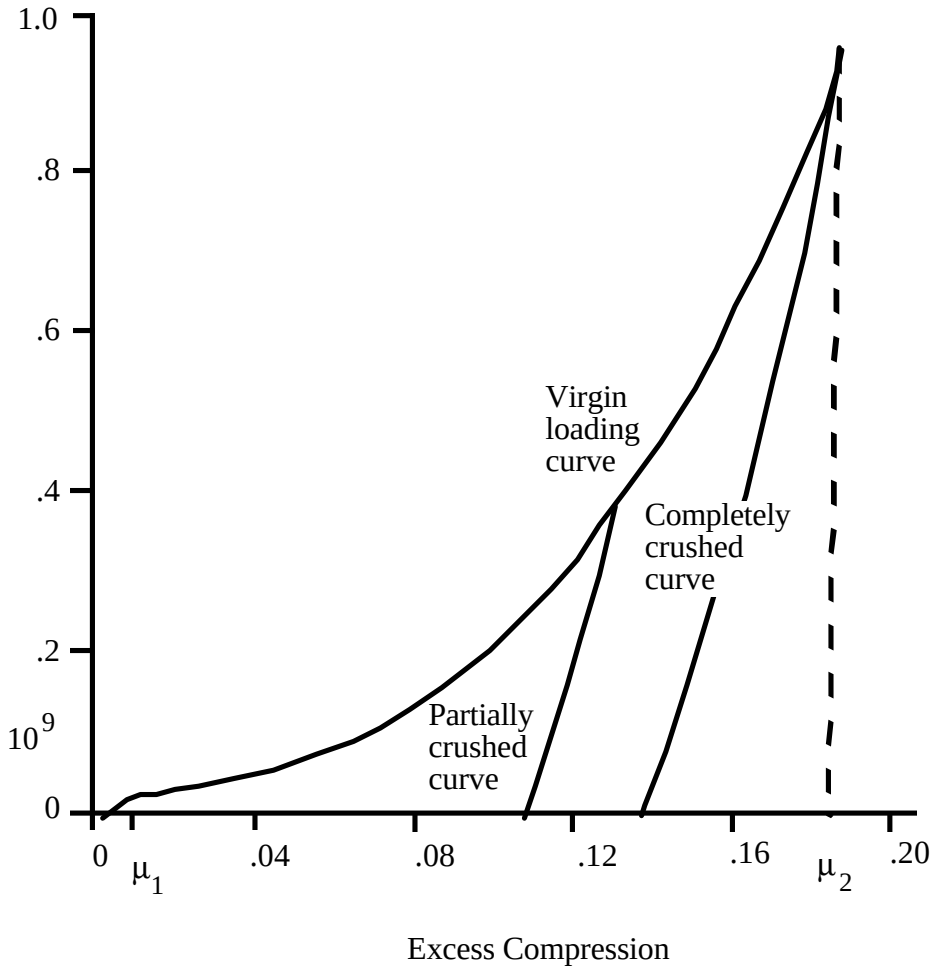


Figure 12.2. Pressure versus compaction curve.

Unloading occurs along the virgin loading curve until the excess compression surpasses μ_1. After that, the unloading follows a path between the completely crushed curve and the virgin loading curve. Reloading will follow this curve back up to the virgin loading curve. Once the excess compression exceeds μ_2, then all unloading will follow the completely crushed curve.

For unloading between μ_1 and μ_2 a partially crushed curve is determined by the relationship:

$$p_{pc}(\mu) = p_{cc}\left(\frac{(1+\mu_B)(1+\mu)}{1+\mu_{\max}} - 1\right).$$

where

$$\mu_B = P_{cc}^{-1}(P_{\max})$$

and the subscripts *pc* and *cc* refer to the partially crushed and completely crushed states, respectively. This is more readily understood in terms of the relative volume (*V*).

$$V = \frac{1}{1+\mu}$$

$$P_{pc}\,(V) = P_{cc}\left(\frac{V_B}{V_{\min}}V\right)$$

This representation suggests that for a fixed $V_{\min}\left(=\frac{1}{\mu_{\max}+1}\right)$ the partially crushed curve will separate linearly from the completely crushed curve as *V* increases to account for pore recovery in the material.

The bulk modulus *K* is determined to be the slope of the current curve times one plus the excess compression:

$$K = \frac{\partial P}{\partial \mu}(1+\mu)$$

The slope $\frac{\partial P}{\partial \mu}$ for the partially crushed curve is obtained by differentiation as:

$$\frac{\partial P}{\partial \mu} = \frac{\partial P_{cc}\left(\frac{(1+\mu_B)(1+\mu)}{(1\mu_{\max})}\right)(1+\mu_B)}{\partial \mu \qquad (1+\mu_{\max})}$$

Simplifying,

$$K = \frac{\partial P_{cc}\left(\mu_a\right)}{\partial \mu}(1+\mu_a)$$

where

$$\mu_a = \frac{(1+\mu_B)(1+\mu)}{(1+\mu_{\max})} - 1.$$

The bulk sound speed is determined from the slope of the completely crushed curve at the current pressure to avoid instabilities in the time step.

The virgin loading and completely crushed curves are modeled with monotonic cubic-splines. An optimized vector interpolation scheme is then used to evaluate the cubic-splines. The bulk modulus and sound speed are derived from a linear interpolation on the derivatives of the cubic-splines.

*HOURGLASS

***HOURGLASS**

Purpose: Define hourglass and bulk viscosity properties. Using the *PART definition this specification is connected to the elements.

Card Format

Card 1	1	2	3	4	5	6	7	8
Variable	HGID	IHQ	QM	IBQ	Q1	Q2	QB	QW
Type	I	I	F	I	F	F	F	F
Default	0	1	.10	0.	1.5	0.06	QM	QM
Remark		1	2&4	3	3	3	4	4

VARIABLE	DESCRIPTION
HGID	Hourglass ID. Unique numbers have to be specified.
IHQ	Hourglass control type. For solid elements five options are available. For quadrilateral shell and membrane lements the hourglass control is based on the formulation of Belytschko and Tsay, i.e., options 1-3 are identical, and options 4-5 are identical: EQ.0:default=1, EQ.1:standard LS-DYNA3D viscous form, EQ.2:Flanagan-Belytschko viscous form, EQ.3: Flanagan-Belytschko viscous form with exact volume integration for solid elements, EQ.4:Flanagan-Belytschko stiffness form, EQ.5:Flanagan-Belytschko stiffness form with exact volume integration for solid elements. A discussion of the hourglass control for shell elements follows at the end of this section.

*HOURGLASS

VARIABLE	DESCRIPTION
QM	Hourglass coefficient. Values of QM that exceed .15 may cause instabilities. The recommended default applies to all options. The stiffness forms, however, can stiffen the response especially if deformations are large and therefore should be used with care. For the shell and membrane elements QM is taken as the membrane hourglass coefficient, the bending as QB, and warping as QW. These coefficients can be specified independently, but generally, QM=QB=QW, is adequate.
IBQ	Bulk viscosity type (See remark 3. below.): EQ.1: standard LS-DYNA3D.
Q2	Quadratic bulk viscosity coefficient.
Q1	Linear bulk viscosity coefficient.
QB	Hourglass coefficient for shell bending. The default: QB=QM. See remark 4. below.
QW	Hourglass coefficient for shell warping. The default: QB=QW.

Remarks:

1. Viscous hourglass control is recommended for problems deforming with high velocities. Stiffness control is preferable for lower velocities. For solid elements the exact integration provides some advantage for highly distorted elements.

2. For automotive crash the stiffness form of the hourglass control with a coefficient of 0.05 is preferred by many users.

3. Bulk viscosity is necessary to propagate shock waves in solid materials and therefore applies only to solid elements. Generally, the default values are okay except in problems where pressures are very high, larger values may be desirable. In low density foams, it may be necessary to reduce the viscosity values since the viscous stress can be significant. It is not advisable to reduce it by more than an order of magnitude.

4. In part, the computational efficiency of the Belytschko-Lin-Tsay and the under integrated Hughes-Liu shell elements are derived from their use of one-point quadrature in the plane of the element. To suppress the hourglass deformation modes that accompany one-point quadrature, hourglass viscous or stiffness based stresses are added to the physical stresses at the local element level. The discussion of the hourglass control that follows pertains to all one point quadrilateral shell and membrane elements in LS-DYNA3D. .

The hourglass shape vector τ_I is defined as

$$\tau_I = h_I - \left(h_J \hat{x}_{aJ}\right) B_{aI}$$

where, $\hat{x}_{aJ}$ are the element coordinates in the local system at the Ith element node, B_{aI} is the strain displacement matrix, and hourglass basis vector is:

$$h = \begin{bmatrix} +1 \\ -1 \\ +1 \\ -1 \end{bmatrix}$$

is the basis vector that generates the deformation mode that is neglected by one-point quadrature. In the above equations and the reminder of this subsection, the Greek subscripts have a range of 2, e.g., $\hat{x}_{aI} = \left(\hat{x}_{1I}, \hat{x}_{2I}\right) = \left(\hat{x}_I, \hat{y}_I\right)$.

The hourglass shape vector then operates on the generalized displacements to produce the generalized hourglass strain rates

$$\dot{q}_\alpha^M = \tau_I \hat{\upsilon}_{\alpha I}$$

$$\dot{q}_\alpha^B = \tau_I \hat{\theta}_{\alpha I}$$

$$\dot{q}_3^W = \tau_I \hat{\upsilon}_{zI}$$

where the superscripts M, B, and W denote membrane, bending, and warping modes, respectively. The corresponding hourglass stress rates are then given by

$$\dot{Q}_\alpha^M = \frac{QM \cdot EtA}{8} B_{\beta I} B_{\beta I} \dot{q}_\alpha^M$$

$$\dot{Q}_\alpha^B = \frac{QB \cdot Et^3 A}{192} B_{\beta I} B_{\beta I} \dot{q}_\alpha^B$$

$$\dot{Q}_3^W = \frac{QW \cdot \kappa G t^3 A}{12} B_{\beta I} B_{\beta I} \dot{q}_3^B$$

where t is the shell thickness. The hourglass coefficients: QM, QB, and QW are generally assigned values between 0.05 and 0.10.

Finally, the hourglass stresses which are updated using the time step, Δt, from the stress rates in the usual way, i.e.,

$$Q^{n+1} = Q^n + \Delta t \dot{Q}$$

and the hourglass resultant forces are then

$$\hat{f}_{\alpha I}^{H} = \tau_I Q_\alpha^M$$

$$\hat{m}_{\alpha I}^{H} = \tau_I Q_\alpha^B$$

$$\hat{f}_{3I}^{H} = \tau_I Q_3^W$$

where the superscript H emphasizes that these are internal force contributions from the hourglass deformations.

*INCLUDE

***INCLUDE**

Purpose: File to be included in this keyword file. The file contents are placed directly at the location of the *INCLUDE line.

Card Format

Card 1 1

Variable	FILENAME
Type	C

VARIABLE	DESCRIPTION
FILENAME	File name of file to be included in this keyword file, 80 characters maximum.

To make the input file easy to maintain, this keyword allows the input file to be split into subfiles. Each subfile can again be split into sub-subfiles and so on. This option is beneficial when the input data deck is very large. Consider the following example:

```
*TITLE
full car model
*INCLUDE
carfront.k
*INCLUDE
carback.k
*INCLUDE
occupantcompartment.k
*INCLUDE
dummy.k
*INCLUDE
bag.k
*CONTACT
• • •
*END
```

Note that the command *END terminates the include file.

The carfront.k file can again be subdivided into rightrail.k, leftrail.k, battery.k, wheelhouse.k, shotgun.k, etc.. Each *.k file can include nodes, elements, boundary conditions, initial conditions, and so on.

```
*INCLUDE
rightrail.k
*INCLUDE
leftrail.k
*INCLUDE
battery.k
*INCLUDE
wheelhouse.k
*INCLUDE
shotgun.k
• • •
• • •
*END
```

*INITIAL

The keyword *INITIAL provides a way of initializing velocities and detonation points. The keyword control cards in this section are defined in alphabetical order:

*INITIAL_DETONATION
*INITIAL_MOMENTUM
*INITIAL_STRESS_BEAM
*INITIAL_STRESS_SHELL
*INITIAL_STRESS_SOLID
*INITIAL_TEMPERATURE_OPTION

Two mutually exclusive methods are available for initial velocity generation:

*INITIAL_VELOCITY
*INITIAL_VELOCITY_NODE

and:

*INITIAL_VELOCITY_GENERATION

The latter is convenient for specifying initial rotational velocities about arbitrary axes. These method for velocity generation must not be mixed in a single input deck.

*INITIAL

*INITIAL_DETONATION

Purpose: Define points to initiate the location of high explosive detonations.

Card Format

Card 1

Variable	PID	X	Y	Z	LT			
Type	I	F	F	F	F			
Default	all HE	0.	0.	0.	0,			

Optional card required if and only if PID=-1.

Card 2

Variable	PEAK	DECAY	XS	YS	ZS	NID		
Type	F	F	F	F	F	I		
Remark	1	1						

VARIABLE	DESCRIPTION
PID	Part ID of high explosive material to be lit, see *PART. However, two other options are available: EQ.-1: an acoustic boundary,.also *BOUNDARY_USA_SURFACE, EQ. 0: all high explosive materials are considered.
X	x-coordinate of detonation point, see Figure 15.1.
Y	y-coordinate of detonation point.
Z	z-coordinate of detonation point.
LT	Lighting time for detonation point. This time is ignored for an acoustic boundary.

VARIABLE	DESCRIPTION
PEAK	Peak pressure, p_o, of incident pressure pulse, see remark below.
DECAY	Decay constant, τ
XS	x-coordinate of standoff point, see Figure 15.1.
YS	y-coordinate of standoff point
ZS	z-coordinate of standoff point
NID	Reference node ID near structure

Remark:

1. The pressure versus time curve is defined by: $p(t) = p_o e^{-\frac{t}{\tau}}$.

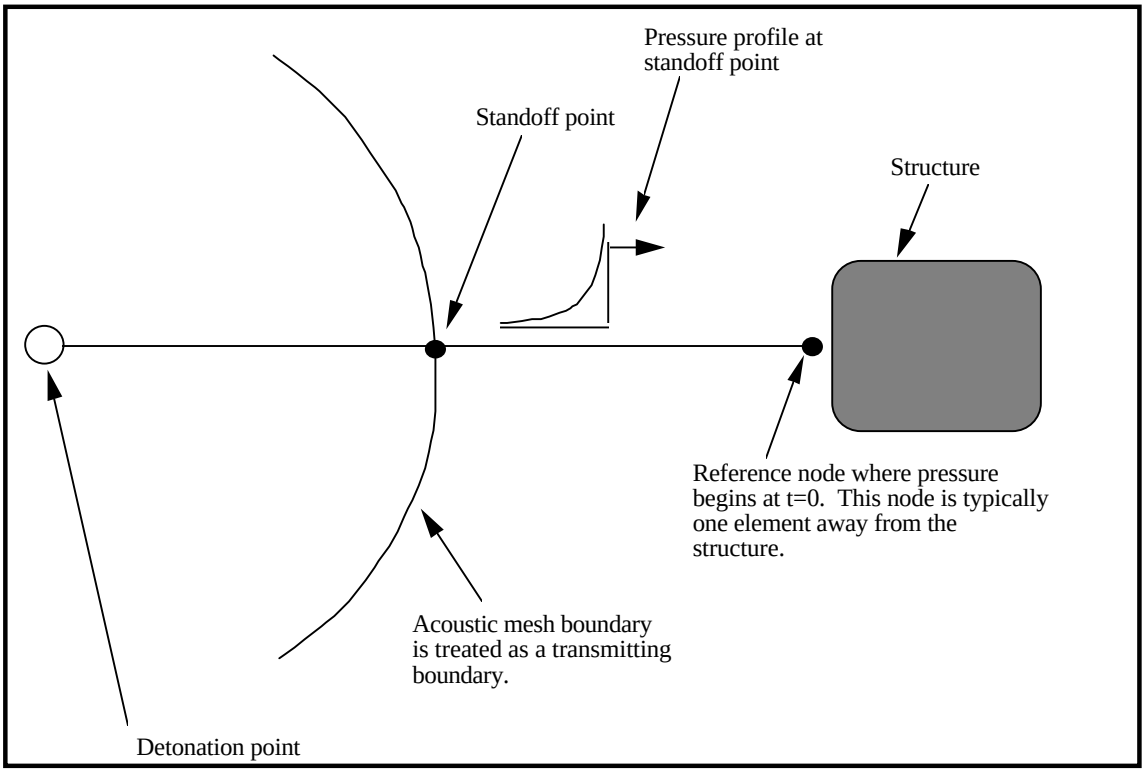

Pressure profile at standoff point
Standoff point
Structure
Reference node where pressure begins at t=0. This node is typically one element away from the structure.
Acoustic mesh boundary is treated as a transmitting boundary.
Detonation point

Figure 15.1 Initialization of the initial pressures due to an explosive disturbance is performed in the acoustic media. LS-DYNA3D automatically determines the acoustic mesh boundary and applies the pressure time history to the boundary. This option is only applicable to the acoustic element formulation, see *SECTION_SOLID.

*INITIAL_MOMENTUM

Purpose: Define initial momentum to be deposited in solid elements. This option is to crudely simulate an impulsive type of loading.

Card Format

	1	2	3	4	5	6	7	8
Variable	EID	MX	MY	MZ	DEPT			
Type	I	F	F	F	F			
Default	none	0.	0.	0.	0,			

VARIABLE	DESCRIPTION
EID	Element ID
MX	Initial x-momentum
MY	Initial y-momentum
MZ	Initial z-momentum
DEPT	Deposition time

***INITIAL_STRESS_BEAM**

Purpose: Initialize stresses and plastic strains in the Hughes-Liu beam elements.

Define as many beams in this section as desired. The input is assumed to terminate when a new keyword is detected.

Card Format

Card 1	1	2	3	4	5	6	7	8
Variable	EID	RULE	NPTS					
Type	I	I	I					
Default	none	none	none					

Define NTPS cards below, one per integration point.

Card 2,...	1	2	3	4	5	6	7	8
Variable	SIG11	SIG22	SIG33	SIG12	SIG23	SIG31	EPS	
Type	F	F	F	F	F	F	F	

VARIABLE	DESCRIPTION
EID	Element ID
RULE	Integration rule type number: EQ.1.0: truss element or discrete beam element, EQ.2.0: 2×2 Gauss quadrature (default beam), EQ.3.0: 3×3 Gauss quadrature, EQ.4.0: 3×3 Lobatto quadrature, EQ.5.0: 4×4 Gauss quadrature.

VARIABLE	DESCRIPTION
NPTS	Number of integration points output.
SIGIJ	Define the IJ stress component.
EPS	Effective plastic strain

***INITIAL_STRESS_SHELL**

Purpose: Initialize stresses and plastic strains for shell elements.

Define as many shell elements in this section as desired. The input is assumed to terminate when a new keyword is detected. It is not necessary for the location of the through thickness integration points to match those used in the elments which are initialized. The data will be interpolated by LS-DYNA3D.

Card Format

Card 1	1	2	3	4	5	6	7	8
Variable	EID	NPLANE	NTHICK					
Type	I	I	I					
Default	none	none	none					

Define NPLANE X NTHICK cards below, one per integration point. For each through thickness point define NPLANE points. NPLANE should be either 1 or 4 corresponding to either 1 or 4 Gauss integration points. If four integration points are specified, they should be ordered such that their in plane parametric coordinates are at:

$$\left(-\frac{\sqrt{3}}{3},-\frac{\sqrt{3}}{3}\right), \quad \left(\frac{\sqrt{3}}{3},-\frac{\sqrt{3}}{3}\right), \quad \left(\frac{\sqrt{3}}{3},\frac{\sqrt{3}}{3}\right), \quad \left(-\frac{\sqrt{3}}{3},\frac{\sqrt{3}}{3}\right),$$

respectively.

Card 2,...	1	2	3	4	5	6	7	8
Variable	T	SIG11	SIG22	SIG33	SIG12	SIG23	SIG31	EPS
Type	F	F	F	F	F	F	F	F

VARIABLE	DESCRIPTION
EID	Element ID
NPLANE	Number of in plane integration points being output.
NTHICK	Number of through thickness integration points.
T	Parametric coordinate of through thickness integration point. between -1 and 1 inclusive.
SIGIJ	Define the IJ stress component.
EPS	Effective plastic strain

***INITIAL_STRESS_SOLID**

Purpose: Initialize stresses and plastic strains for solid elements.

Define as many solid elements in this section as desired. The input is assumed to terminate when a new keyword is detected. If eight points are defined for 1 point LS-DYNA3D solid elements, the average value will be taken.

Card Format

Card 1

1	2	3	4	5	6	7	8
Variable	EID	NINT						
Type	I	I						
Default	none	none						

Define NINT cards below, one per integration point. NINT should be either 1 or 8. If eight Gauss integration points are specified, they should be ordered such that their parametric coordinates are located at:

$$\left(-\frac{\sqrt{3}}{3},-\frac{\sqrt{3}}{3},-\frac{\sqrt{3}}{3}\right),\left(\frac{\sqrt{3}}{3},-\frac{\sqrt{3}}{3},-\frac{\sqrt{3}}{3}\right),\quad\left(\frac{\sqrt{3}}{3},\frac{\sqrt{3}}{3},-\frac{\sqrt{3}}{3}\right),\quad\left(-\frac{\sqrt{3}}{3},\frac{\sqrt{3}}{3},-\frac{\sqrt{3}}{3}\right),$$

$$\left(-\frac{\sqrt{3}}{3},-\frac{\sqrt{3}}{3},\frac{\sqrt{3}}{3}\right),\quad\left(\frac{\sqrt{3}}{3},-\frac{\sqrt{3}}{3},\frac{\sqrt{3}}{3}\right),\quad\left(\frac{\sqrt{3}}{3},\frac{\sqrt{3}}{3},\frac{\sqrt{3}}{3}\right),\quad\left(-\frac{\sqrt{3}}{3},\frac{\sqrt{3}}{3},\frac{\sqrt{3}}{3}\right),$$

respectively.

Card 2,...

1	2	3	4	5	6	7	8
Variable	SIG11	SIG22	SIG33	SIG12	SIG23	SIG31	EPS	
Type	F	F	F	F	F	F	F	

VARIABLE	DESCRIPTION
EID	Element ID
NINT	Number of integration points either 1 or 8.
SIGIJ	Define the IJ stress component.
EPS	Effective plastic strain

INITIAL_TEMPERATURE_*OPTION

Available options are:

NODE
SET

Purpose: Define initial nodal point temperatures using nodal set IDÕs or node numbers. These initial temperatures are used in a thermal only analysis or a coupled thermal/structural analysis. See also *CONTROL_THERMAL_SOLVER, *CONTROL_THERMAL_TIMESTEP, and CONTROL_ THERMAL_NONLINEAR.

For thermal loading in a structural only analysis, see *LOAD_THERMAL_option.

Card Format

Card 1	1	2	3	4	5	6	7	8
Variable	NSID/NID	TEMP						
Type	I	I						
Default	none	0.						
Remark	1							

VARIABLE	DESCRIPTION
NSID/NID	Nodal set ID or nodal point ID, see also *SET_NODES: EQ.0: all nodes are included (set option only).
TEMP	Temperature at node or node set.

Remark:

1. If a nodal temperature is specified on more than one input card, then the last set input will determine its temperature unless it is specified on a *INITIAL_TEMPERATURE_NODE card.

*INITIAL_VELOCITY

Purpose: Define initial nodal point translational velocities using nodal set IDÕs. This may also be used for sets in which some nodes have other velocities. See NSIDEX below.

Card Format

Card 1

Variable	NSID	NSIDEX	BOXID					
Type	I	I	I					
Default	none	0.	0.					
Remark	1							

Card 2

Variable	VX	VY	VZ	VXR	VYR	VZR		
Type	F	F	F	F	F	F		
Default	0.	0.	0.	0.	0.	0.		

Define the following card if and only if NSIDEX>0.

Variable	VXE	VYE	VZE	VXRE	VYRE	VZRE		
Type	F	F	F	F	F	F		
Default	0.	0.	0.	0.	0.	0.		

VARIABLE	DESCRIPTION
NSID	Nodal set ID, see *SET_NODES, containing nodes for initial velocity: EQ.0: all nodes are included.
NSIDEX	Nodal set I, see *SET_NODES, containing nodes that are exempted from the imposed velocities and may have other initial velocities.
BOXID	All nodes in box which belong to NSID are initialized. Nodes outside the box are not initialized. Exempted nodes are initialized to velocities defined by VXE, VYE, and VZE below regardless of their location relative to the box.
VX	Initial velocity in x-direction
VY	Initial velocity in y-direction
VZ	Initial velocity in z-direction
VXR	Initial rotational velocity about the x-axis
VYR	Initial rotational velocity about the y-axis
VZR	Initial rotational velocity about the z-axis
VXE	Initial velocity in x-direction of exempted nodes
VYE	Initial velocity in y-direction of exempted nodes
VZE	Initial velocity in z-direction of exempted nodes
VXRE	Initial rotational velocity in x-direction of exempted nodes
VYRE	Initial rotational velocity in y-direction of exempted nodes
VZRE	Initial rotational velocity in z-direction of exempted nodes

Remarks:

1. This generation input must not be used with *INITIAL_VELOCITY_GENERATION keyword.

2. If a node is initialized on more than one input card set, then the last set input will determine its velocity. However, if the nodal velocity is also specified on a *INITIAL_VELOCITY_NODE card, then the velocity specification on this card will be used.

*INITIAL_VELOCITY_NODE

Purpose: Define initial nodal point velocities for a node.

Card Format

	1	2	3	4	5	6	7	8
Variable	NID	VX	VY	VZ	VXR	VYR	VZR	
Type	I	F	F	F	F	F	F	
Default	none	0.	0.	0.	0.	0.	0.	

VARIABLE	DESCRIPTION
NID	Node ID
VX	Initial translational velocity in x-direction
VY	Initial translational velocity in y-direction
VZ	Initial translational velocity in z-direction
VXR	Initial rotational velocity about the x-axis
VYR	Initial rotational velocity about the y-axis
VZR	Initial rotational velocity about the z-axis

See remark on *INITIAL_VELOCITY card.

***INITIAL_VELOCITY_GENERATION**

Purpose: Define initial velocities for rotating and translating bodies.

Card Format

Card 1	1	2	3	4	5	6	7	8
Variable	SID	STYP	OMEGA	VX	VY	VZ		
Type	I	I	F	F	F	F		
Default	none	none	0.	0.	0.	0.		

Card 2	1	2	3	4	5	6	7	8
Variable	XC	YC	ZC	NX	NY	NZ	PHASE	
Type	F	F	F	F	F	F	I	
Default	0.	0.	0.	0.	0.	0.	0	

VARIABLE	DESCRIPTION
SID	Set ID, if zero STYP is ignored and all velocities are set.
STYP	Set type: EQ.1: part set ID, see *SET_PART, EQ.2: part ID, see *PART, EQ.3: node set ID, see *SET_NODE.
OMEGA	Angular velocity about rotational axis
VX	Initial translational velocity in global x-direction.
VY	Initial translational velocity in global y-direction.
VZ	Initial translational velocity in global z-direction. .

VARIABLE	DESCRIPTION
XC	x-coordinate on rotational axis
YC	y-coordinate on rotational axis
ZC	z-coordinate on rotational axis
NX	x-direction cosine
NY	y-direction cosine.
NZ	z-direction cosine.
PHASE	Flag specifying phase of the analysis the velocities apply to: EQ.0. Velocities applied immediately. EQ.1. Velocities applied after dynamic relaxation.

Remarks:

1. This generation input must not be used with *INITIAL_VELOCITY or *INITIAL_VELOCITY_NODE options.

2. The velocities are initialized in the order the *INITIAL_VELOCITY_GENERATION input is defined. Later input via the *INITIAL_VELOCITY_GENERATION keyword may overwrite the velocities previously set.

*INTEGRATION

*INTEGRATION_BEAM

Purpose: Define user defined through the thickness integration rules for the beam element.

Card Format

Card 1	1	2	3	4	5	6	7	8
Variable	IRID	NIP	RA	ICST				
Type	I	I	F	I				
Default	none	0	0.0	0				

Define the following card if and only if ICST>0.

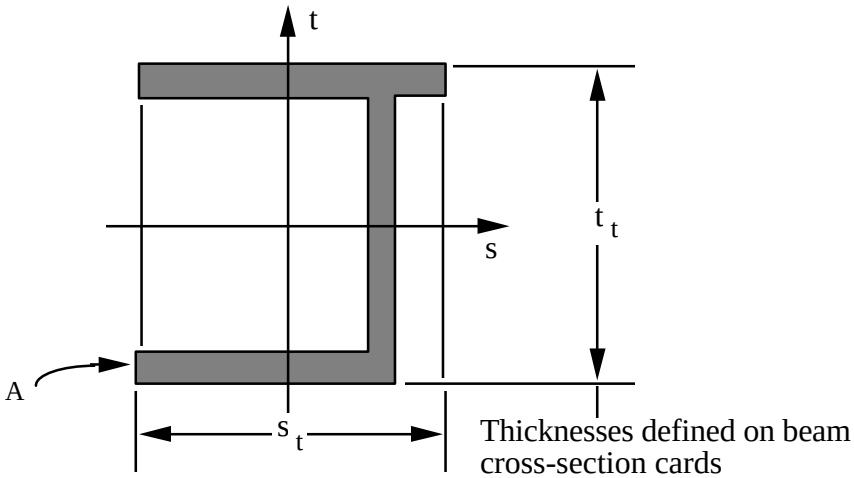
	1	2	3	4	5	6	7	8
Variable	W	TF	D	TW	SREF	TREF		
Type	F	F	F	F	F	F		
Default	none	none	none	none	1.0	0.0		

Define NIP cards below (Skip if NIP=0).

	1	2	3	4	5	6	7	8
Variable	S	T	WF					
Type	F	F	F					

*INTEGRATION

VARIABLE	DESCRIPTION
IRID	Integration rule ID. IRID refers to IRID on *SECTION_BEAM card.
NIP	Number of integration points, see also ICST.
RA	Relative area of cross section, i.e., the actual cross-sectional area divided by the area defined by the product of the specified thickness in the s direction and the thickness in the t direction. See also ICST below and Figure 16.1.
ICST	Standard cross section type, ICST. If this type is nonzero then NIP and the relative area above should be input as zero. See the discussion following the input description Figures 16.3a and 16.3b. EQ.1: W-section, EQ.2: C-section, EQ.3: Angle section, EQ.4: T-section, EQ.5: Rectangular tubing, EQ.6: Z-section,. EQ.7: Trapezoidal section
W	w, flange width
TF	t_f, flange thickness
D	d, depth
TW	t_w, web thickness
SREF	s_{ref}, location of reference surface normal to s, for the Hughes-Liu beam only. This option is only useful if the beam is connected to a shell or another beam on its outer surface, see also *SECTION_BEAM.
TREF	t_{ref}, location of reference surface normal to t, for the Hughes-Liu beam only. This option is only useful if the beam is connected to a shell or another beam on its outer surface, see also *SECTION_BEAM.
S	s coordinate of integration point
T	t coordinate of integration point
WF	Weighting factor, A_{ri} , i.e., the area associated with the integration point divided by actual cross sectional area $A_{ri} = A_i / A$, see Figure 16.2.



$$\text{Relative Area} = \frac{A}{s_t \cdot t_t}$$

Figure 16.1. Definition of relative area for user defined integration rule.

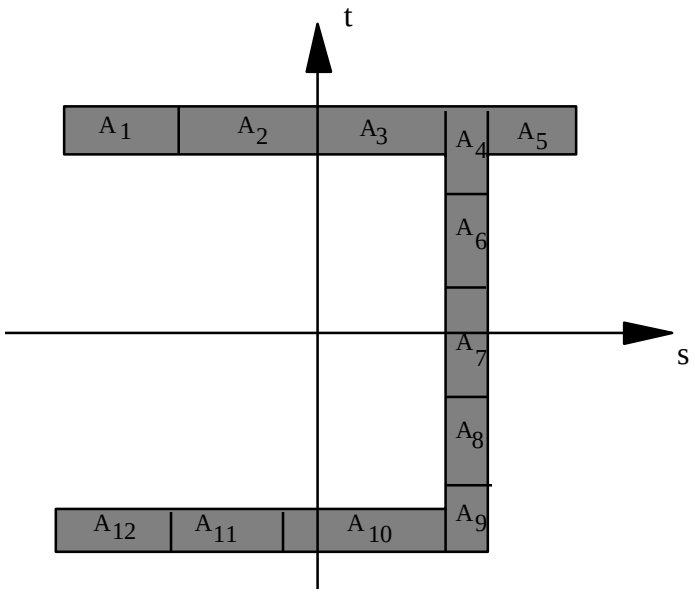


Figure 16.2. Definition of integration points for user defined integration rule.

The input for standard beam section types is defined below. In Figures 16.3a and 16.3b, the dimensions are shown on the left and the location of the integration points are shown on the right. If a quantity is not defined in the sketch, then it should be set to zero in the input. The input quantities include:

w = flange width
t_f = flange thickness
d = depth
t_w = web thickness
s_{ref} = location of reference surface normal to s, Hughes-Liu beam only
t_{ref} = location of reference surface normal to t, Hughes-Liu beam only

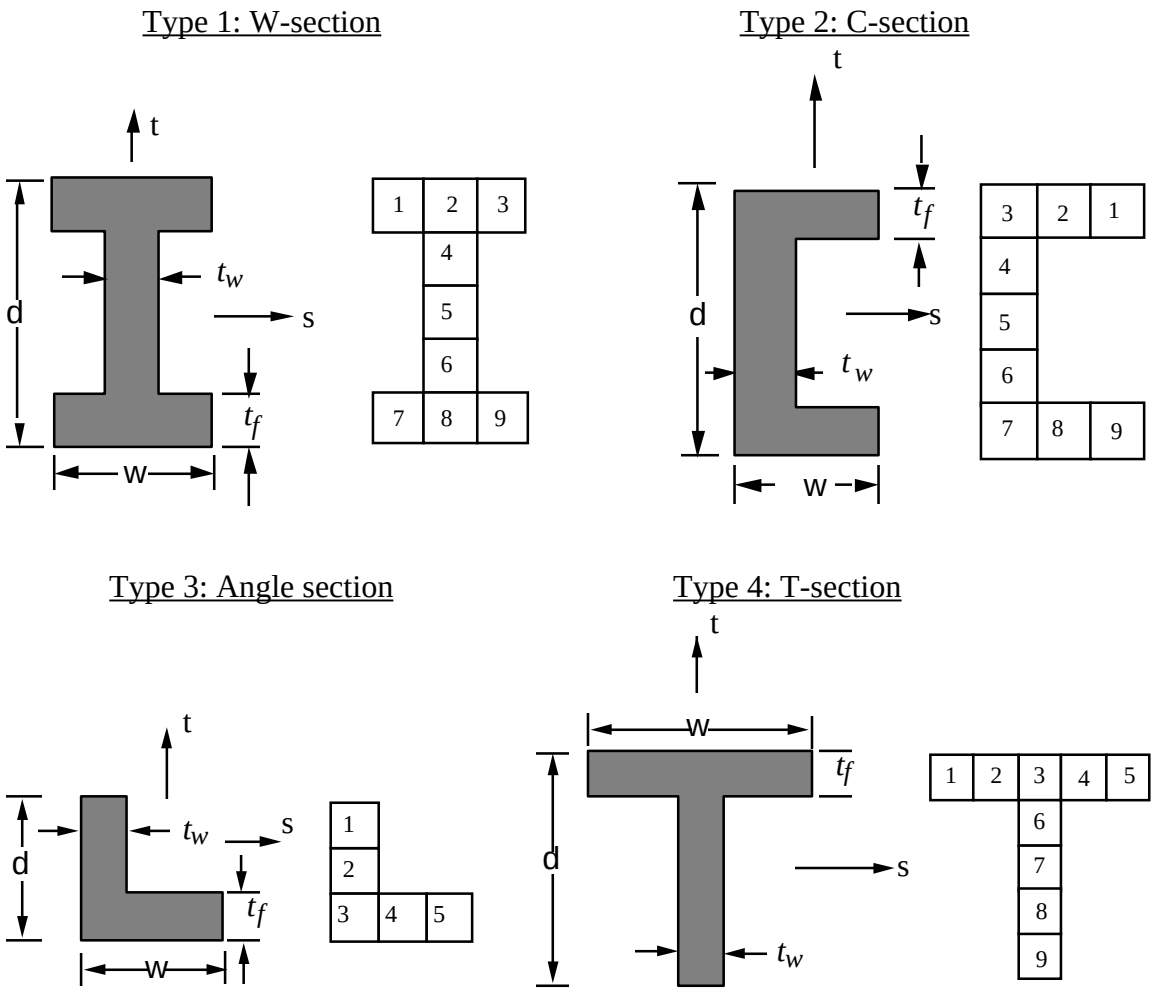
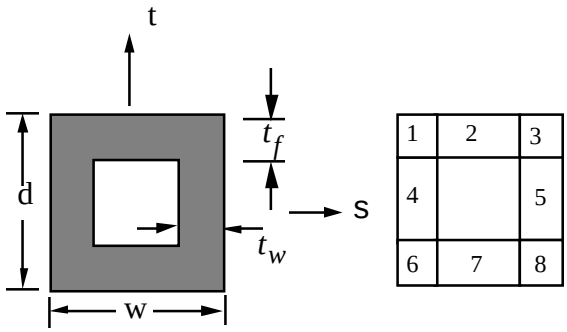
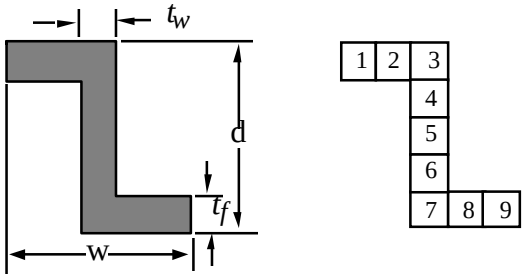


Figure 16.3a. Standard beam cross sections.

Type 5: Rectangular tubing



Type 6: Z-section



Type 7: Trapezoidal section.

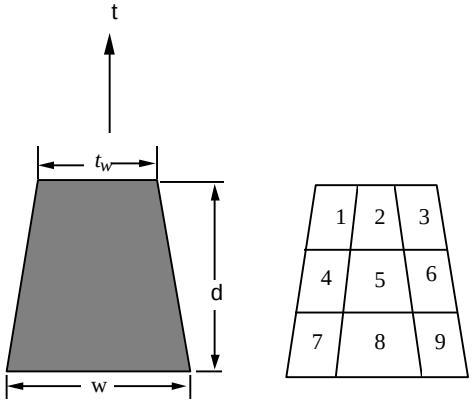


Figure 16.3b. Standard beam cross sections.

*INTEGRATION

*INTEGRATION_SHELL

Purpose: Define user defined through the thickness integration rules for the shell element.

Card Format

Card 1	1	2	3	4	5	6	7	8
Variable	IRID	NIP	ESOP					
Type	I	I	I					

Define NIP cards below.

	1	2	3	4	5	6	7	8
Variable	S	WF	PID					
Type	F	F	I					

VARIABLE	DESCRIPTION
IRID	Integration rule ID (IRID refers to IRID on *SECTION_SHELL card).
NIP	Number of integration points
ESOP	Equal spacing of integration points option: EQ.0: integration points are defined below, EQ.1: integration points are equally spaced through thickness such that the shell is subdivided into NIP layers of equal thickness.
S	Coordinate of integration point in range -1 to 1.
WF	Weighting factor. This is typically the thickness associated with the integration point divided by actual shell thickness, i.e., the weighting factor for the ith integration point = $\frac{\Delta t_i}{t}$ as seen in Figure 16.4. Not necessary if ESOP=1.
PID	Optional part ID if different from the ID specified on the element card. The material type is not allowed to change, see *PART.

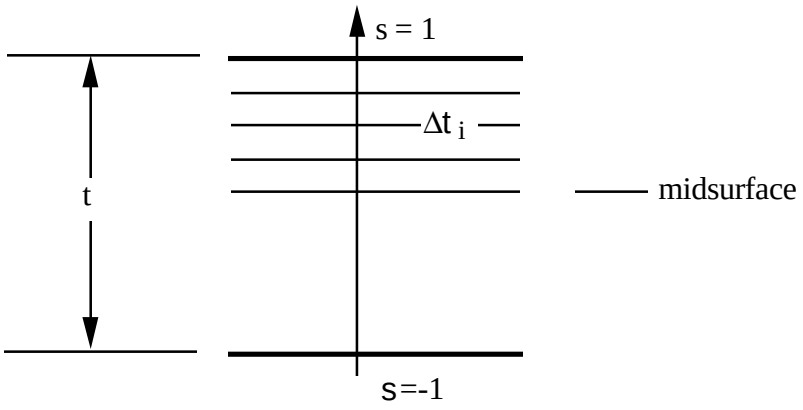


Figure 16.4. In the user defined shell integration rule the ordering of the integration points is arbitrary.

*INTERFACE

***INTERFACE_COMPONENT_OPTION**

Options include:

NODE
SEGMENT

Purpose: Define an interface for linking calculations. This card applies to the first analysis for storing interfaces in the file specified by Z=isf1 on the execution command line.

This capability allows the definition of interfaces that isolate critical components. A database is created that records the motion of the interfaces. In later calculations the isolated components can be reanalyzed with arbitrarily refined meshes with the motion of their boundaries specified by the database created by this input. The interfaces defined here become the masters in the tied interface options.

Each definition consists of a set of cards that define the interface. Interfaces may consists of a set of four node segments for moving interfaces of solid elements, a line of nodes for treating interfaces of shells, or a single node for treating beam and spring elements.

Card Format

	1	2	3	4	5	6	7	8
Variable	SID							
Type	I							

VARIABLE	DESCRIPTION
SID	Set ID, see *SET_NODE or *SET_SEGMENT.

INTERFACE_LINKING_DISCRETE_NODE_*OPTION

Options include:

NODE
SET

Purpose: Define an interface for linking discrete nodes to an interface file. This link applies to spring and beam elements only.

Card Format

	1	2	3	4	5	6	7	8
Variable	NID/NSID	IFID						
Type	I	I						

VARIABLE	DESCRIPTION
NID	Node ID or Node set ID to be moved by interface file, see *NODE or *SET_NODE.
IFID	Interface ID in interface file.

***INTERFACE_LINKING_SEGMENT**

Purpose: Define an interface for linking segments to an interface file for the second analysis using L=isf2 on the execution command line. This applies segments on shell and solid elements.

Card Format

	1	2	3	4	5	6	7	8
Variable	SSID	IFID						
Type	I	I						

VARIABLE	DESCRIPTION
SSID	Segment set to be moved by interface file.
IFID	Interface ID in interface file.

*INTERFACE_LINKING_EDGE

Purpose: Define an interface for linking a series of nodes in shell structure to an interface file for the second analysis using L=isf2 on the execution command line. This link applies segments on shell elements only.

Card Format

	1	2	3	4	5	6	7	8
Variable	NSID	IFID						
Type	I	I						

VARIABLE	DESCRIPTION
NSID	Node set ID to be moved by interface file.
IFID	Interface ID in interface file.

***INTERFACE_JOY**

Purpose: Define an interface for linking calculations by moving a nodal interface.

Card Format

	1	2	3	4	5	6	7	8
Variable	SID							
Type	I							

VARIABLE	DESCRIPTION
SID	Nodal set ID, see *SET_NODE.

*INTERFACE_SPRINGBACK

Purpose: Define a material subset for an implicit springback calculation in LS-NIKE3D and any nodal constraints to eliminate rigid body degrees-of-freedom.

Card Format

	1	2	3	4	5	6	7	8
Variable	PSID							
Type	I							

VARIABLE	DESCRIPTION
PSID	Part set ID for springback, see ∗SET_PART.

Define a list of nodal points that are constrained for the springback. This section is terminated by an "∗" indicating the next input section.

Card Format

	1	2	3	4	5	6	7	8
Variable	NID	TC	RC					
Type	I	F	F					
Default	none	0.	0.					

VARIABLE	DESCRIPTION
NID	Node ID, see *NODE.

VARIABLE	DESCRIPTION
TC	Tranlational Constraint: EQ.0: no constraints, EQ.1: constrained x displacement, EQ.2: constrained y displacement, EQ.3: constrained z displacement, EQ.4: constrained x and y displacements, EQ.5: constrained y and z displacements, EQ.6: constrained z and x displacements.
RC	Rotational constraint: EQ.0: no constraints, EQ.1: constrained x rotation, EQ.2: constrained y rotation, EQ.3: constrained z rotation, EQ.4: constrained x and y rotations, EQ.5: constrained y and z rotations, EQ.6: constrained z and x rotations, EQ.7: constrained x, y, and z rotations.

*LOAD

The keyword *LOAD provides a way of defining applied forces. The keyword control cards in this section are defined in alphabetical order:

*LOAD_BEAM_OPTION
*LOAD_BODY_OPTION
*LOAD_BODY_GENERALIZED
*LOAD_BRODE
*LOAD_DENSITY_DEPTH
*LOAD_HEAT_GENERATION_OPTION
*LOAD_NODE_OPTION
*LOAD_RIGID_BODY
*LOAD_SEGMENT
*LOAD_SEGMENT_SET
*LOAD_SHELL_OPTION
*LOAD_SUPERPLASTIC_FORMING
*LOAD_THERMAL_OPTION
*LOAD_THERMAL_CONSTANT
*LOAD_THERMAL_CONSTANT_NODE
*LOAD_THERMAL_LOAD_CURVE
*LOAD_THERMAL_TOPAZ
*LOAD_THERMAL_VARIABLE
*LOAD_THERMAL_VARIABLE_NODE

***LOAD_BEAM_OPTION**

Options include:

ELEMENT
SET

Purpose: Apply the distributed traction load along any local axis of beam or a set of beams. The local axes are defined in Figure 18.1, see also *ELEMENT_BEAM.

Card Format

	1	2	3	4	5	6	7	8
Variable	option	DAL	LCID	SF				
Type	I	I	I	F				
Default	none	none	none	1.				
Remarks								

VARIABLE	DESCRIPTION
EID/ESID	Beam ID (EID) or beam set ID (ESID), see *ELEMENT_BEAM or *SET_ BEAM.
DAL	Direction of applied load: EQ.1: along r-axis of beam, EQ.2: along s-axis of beam, EQ.3: along t-axis of beam.
LCID	Load curve ID, see *DEFINE_CURVE.
SF	Load curve scale factor. This is for a simple modification of the function values of the load curve.

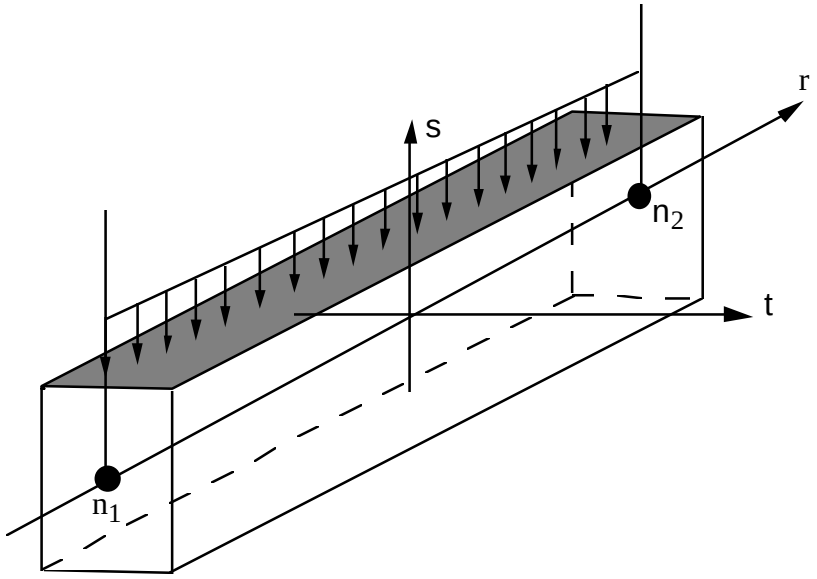


Figure 18.1. Applied traction loads are given in force per unit length. The s and t directions are defined on the *ELEMENT_BEAM keyword.

***LOAD_BODY_OPTION**

Options incude for base accelerations:

X
Y
Z

for angular velocities:

RX
RY
RZ

and to specifiy a part set:

PARTS

Purpose: Define body force loads due to a prescribed base acceleration or angular velocity using global axes directions. This data applies to all nodes in the complete problem unless a part subset is specified via the *LOAD_BODY_PARTS keyword. If a part subset is defined then all nodal points belonging to the subset will have body forces applied. The parts specified via the *LOAD_ BODY_PARTS keyword apply to the options X, Y, Z, RX, RY, and RZ above , i.e., different part sets may not apply to different options. Only one part set is expected.

Card Format for options: X, Y, Z, RX, RY, and RZ.

	1	2	3	4	5	6	7	8
Variable	LCID	SF	LCIDDR	XC	YC	ZC		
Type	I	F	I	F	F	F		
Default	none	1.	0	0.	0.	0.		

Card Format for option: PARTS.

	1	2	3	4	5	6	7	8
Variable	PSID							
Type	I							
Default	none							

VARIABLE	DESCRIPTION
LCID	Load curve ID, see *DEFINE_CURVE.
SF	Load curve scale factor
LCIDDR	Load curve ID for dynamic relaxation phase (optional). This is only needed if dynamic relaxation is defined. See *CONTROL_DYNAMIC_RELAX-ATION
XC	X-center of rotation, define for angular velocities.
YC	Y-center of rotation, define for angular velocities.
ZC	Z-center of rotation, define for angular velocities.
PSID	Part set ID.

Remark:

1. Angular velocity is assumed to have the units of radians per unit time about a global axis.

*LOAD_BODY_GENERALIZED

Purpose: Define body force loads due to a prescribed base acceleration or a prescribed angular velocity over a subset of the complete problem. The subset is defined by using nodes.

Card Format

Card 1

Variable	N1	N2	LCID	DRLCID	XC	YC	ZC	
Type	I	I	I	I	F	F	F	
Default	none	none	none	0	0.	0.	0.	
Remarks					1	1	1	

Card 2

Variable	AX	AY	AZ	OMX	OMY	OMZ		
Type	F	F	F	F	F	F		
Default	0.	0.	0.	0.	0.	0.		
Remarks								

VARIABLE	DESCRIPTION
N1	Beginning node ID for body force load.
N2	Ending node ID for body force load.
LCID	Load curve ID, see *DEFINE_CURVE.
DRLCID	Load curve ID for dynamic relaxation phase. Only necessary if dynamic relaxation is defined. See *CONTROL_DYNAMIC_RELAXATION.

VARIABLE	DESCRIPTION
XC	X-center of rotation. Define only for angular velocity.
YC	Y-center of rotation. Define only for angular velocity.
ZC	Z-center of rotation. Define only for angular velocity.
AX	Scale factor for acceleration in x-direction
AY	Scale factor for acceleration in y-direction
AZ	Scale factor for acceleration in z-direction
OMX	Scale factor for x-angular velocity
OMY	Scale factor for y-angular velocity
OMZ	Scale factor for z-angular velocity

Remark:

1 Required for angular velocity loading.

*LOAD_BRODE

Purpose: Define Brode function for application of pressure loads due to explosion, see Brode [1970], also see *LOAD_SEGMENT, *LOAD_SEGMENT_SET, or LOAD_SHELL.

Card Format

Card 1	1	2	3	4	5	6	7	8
Variable	YLD	BHT	XBO	YBO	ZBO	TBO	TALC	SFLC
Type	F	F	F	F	F	F	I	I
Default	0.0	0.0	0.0	0.0	0.0	0.0	0	0
Remarks							1	1

Card 2	1	2	3	4	5	6	7	8
Variable	CFL	CFT	CFP					
Type	F	F	F					
Default	0.0	0.0	0.0					

VARIABLE	DESCRIPTION
YLD	Yield (Kt)
BHT	Height of burst
XBO	x-coordinates of Brode origin
YBO	y-coordinates of Brode origin
ZBO	z-coordinates of Brode origin
TBO	Time offset of Brode origin

VARIABLE	DESCRIPTION
TALC	Load curve number giving time of arrival versus range relative to Brode origin (space, time), see *DEFINE_CURVE and remark below.
SFLC	Load curve number giving yield scaling versus scaled time (time relative to Brode origin divided by [yield$^{(**1Ú3)}$])origin (space, time), see *DEFINE_ CURVE and remark below.
CFL	Conversion factor - kft to LS-DYNA3D length units.
CFT	Conversion factor - milliseconds to LS-DYNA3D time units.
CFP	Conversion factor - psi to LS-DYNA3D pressure units.

Remark:

1. If these curves are defined a variable yield is assumed. Both load curves must be specified for the variable yield option. If this option is used, the shock time of arrival is found from the time of arrival curve. The yield used in the Brode formulas is computed by taking the value from the yield scaling curve at the current time/[yield$^{(**1Ú3)}$] and multiplying that value by yield.

*LOAD_DENSITY_DEPTH

Purpose: Define density versus depth for gravity loading. This option has been occasionally used for analyzing underground and submerged structures where the gravitational preload is important.

Only one load curve and direction is permitted. If multiple cards are used LCID and DIR should not change.

Card Format

	1	2	3	4	5	6	7	8
Variable	PSID	GC	DIR	LCID				
Type	I	F	I	I				
Default	0.0	0.0	1	0				

VARIABLE	DESCRIPTION
PID	Part set ID, see *PART or *SET_PART_OPTION.
GC	Gravitational acceleration value.
DIR	Direction of loading: EQ.1: global x, EQ.2: global y, EQ.3: global z.
LCID	Load curve ID defining density versus depth, see *DEFINE_CURVE.

LOAD_HEAT_GENERATION_*OPTION

Available options are:

SET
SOLID

Purpose: Define solid elements or solid element set with heat generation.

Card Format

	1	2	3	4	5	6	7	8
Variable	SID	LCID	CMULT					
Type	I	I	F					
Default	none	none	1.0					

VARIABLE	DESCRIPTION
SID	Solid element set ID or solid element ID, see *SET_SOLID or *ELEMENT_SOLID, respectively.
LCID	Load curve ID for volumetric heat generation rate, $\dot{q}'''$: GT.0: function versus time, EQ.0: use multiplier value CMULT only, LT.0: function versus temperature.
CMULT	Curve multiplier for $\dot{q}'''$. Depending on the definition of LCID this value is either used for scaling or for constant heat generation.

*LOAD

LOAD_NODE_*OPTION

Options include:

POINT
SET

Purpose: Apply a concentrated nodal force to a node or a set of nodes.

Card Format

	1	2	3	4	5	6	7	8
Variable	NODE/NSID	DOF	LCID	SF	CID	M1	M2	M3
Type	I	I	I	F	I	I	I	I
Default	none	none	none	1.	0	0	0	0
Remarks					1	2		

VARIABLE	DESCRIPTION
NODE/NSID	Node ID or nodal set ID (NSID), see *SET_NODE_OPTION.
DOF	Applicable degrees-of-freedom: EQ.1: x-direction of load action, EQ.2: y-direction of load action, EQ.3: z-direction of load action, EQ.4: follower force, see remark 2 on next page, EQ.5: moment about the x-axis, EQ.6: moment about the y-axis, EQ.7: moment about the z-axis.
LCID	Load curve ID, see *DEFINE_CURVE.
SF	Load curve scale factor.
CID	Coordinate system ID (optional), see remark 1 on next page.

VARIABLE	DESCRIPTION
M1	Node 1 ID. Only necessary if DOF.EQ.4, see remark 2 below.
M2	Node 2 ID. Only necessary if DOF.EQ.4, see remark 2 below.
M3	Node 3 ID. Only necessary if DOF.EQ.4, see remark 2 below.

Remarks:

1 The global coordinate system is the default. The local coordinate system IDÕs are defined in the *DEFINE_COORDINATE_SYSTEM section.

2. Nodes M_1, M_2, M_3 must be defined for a follower force. The follower force acts normal to the plane defined by these nodes as depicted in Figure 18.2. The positive t-direction is found by the cross product $t = v \times w$ where v and w are vectors as shown.

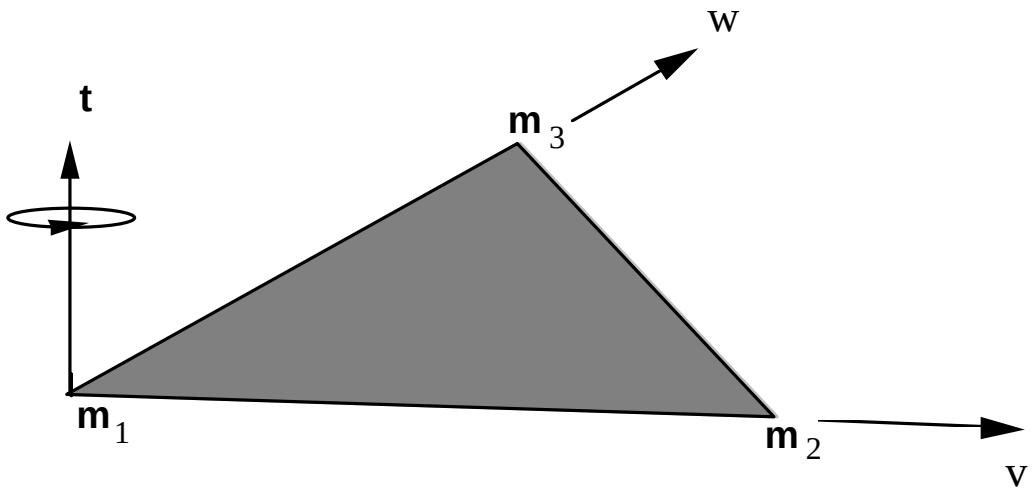


Figure 18.2. Follower force acting on plane defined by nodes m_1, m_2, and m_3. In this case, the load is applied to node m_1; i.e., $m=m_1$. Positive force acts in the positive t-direction.

*LOAD_RIGID_BODY

Purpose: Apply a concentrated nodal force to a rigid body. The force is applied at the cener of mass or a moment is applied around a global axis. As an option local axes can be defined for force or moment directions.

Card Format

	1	2	3	4	5	6	7	8
Variable	PID	DOF	LCID	SF	CID	M1	M2	M3
Type	I	I	I	F	I	I	I	I
Default	none	none	none	1.	0	0	0	0
Remark					1	2		

VARIABLE	DESCRIPTION
PID	Part ID of the rigid body, see *PART_OPTION.
DOF	Applicable degrees-of-freedom: EQ.1: x-direction of load action, EQ.2: y-direction of load action, EQ.3: z-direction of load action, EQ.4: follower force, see remark 2 on next page, EQ.5: moment about the x-axis, EQ.6: moment about the y-axis, EQ.7: moment about the z-axis.
LCID	Load curve ID, see *DEFINE_CURVE.
SF	Load curve scale factor
CID	Coordinate system ID
M1	Node 1 ID. Only necessary if DOF.EQ.4, see remark 2 on next page.
M2	Node 2 ID. Only necessary if DOF.EQ.4, see remark 2 on next page.
M3	Node 3 ID. Only necessary if DOF.EQ.4, see remark 2 on next page.

Remarks:

1 The global coordinate system is the default. The local coordinate system IDÕs are defined in the *DEFINE_COORDINATE_SYSTEM section. This local axis is fixed in inertial space, i.e., it does not move with the rigid body.

2. Nodes M_1, M_2, M_3 must be defined for a follower force. The follower force acts normal to the plane defined by these nodes as depicted in Figure 18.2. The positive t-direction is found by the cross product $t = v \times w$ where v and w are vectors as shown. The follower force is applied at the center of mass.

*LOAD_SEGMENT

Purpose: Apply the distributed pressure load over one triangular or quadrilateral segment defined by four nodes. The pressure convention follows Figure 18.3.

Card Format

	1	2	3	4	5	6	7	8
Variable	LCID	SF	AT	N1	N2	N3	N4	
Type	I	F	F	I	I	I	I	
Default	none	1.	0.	none	none	none	none	
Remarks	1	2		3				

VARIABLE	DESCRIPTION
LCID	Load curve ID, see *DEFINE_CURVE.
SF	Load curve scale factor
AT	Arrival time for pressure or birth time of pressure.
N1	Node Number
N2	Node Number
N3	Node Number
N4	Node Number

Remarks:

1 If LCID is input as -1, then the Brode function is used to determine the pressure for the segments, see *LOAD_BRODE.

2. The load curve multipliers may be used to increase or decrease the pressure. The time value is not scaled.

3. Triangular segments are defined by repeating the third node.

***LOAD_SEGMENT_SET**

Purpose: Apply the distributed pressure load over each segment in a segment set. The pressure convention follows Figure 18.3.

Card Format

	1	2	3	4	5	6	7	8
Variable	SSID	LCID	SF	AT				
Type	I	I	F	F				
Default	none	none	1.	0.				
Remarks		1	2					

VARIABLE	DESCRIPTION
SSID	Segment set ID, see *SET_SEGMENT.
LCID	Load curve ID, see *DEFINE_CURVE.
SF	Load curve scale factor
AT	Arrival time for pressure or birth time of pressure.

Remarks:

1 If LCID is input as -1, then the Brode function is used to determine pressure for the segment set, also see *LOAD_BRODE.

2. The load curve multipliers may be used to increase or decrease the pressure. The time value is not scaled.

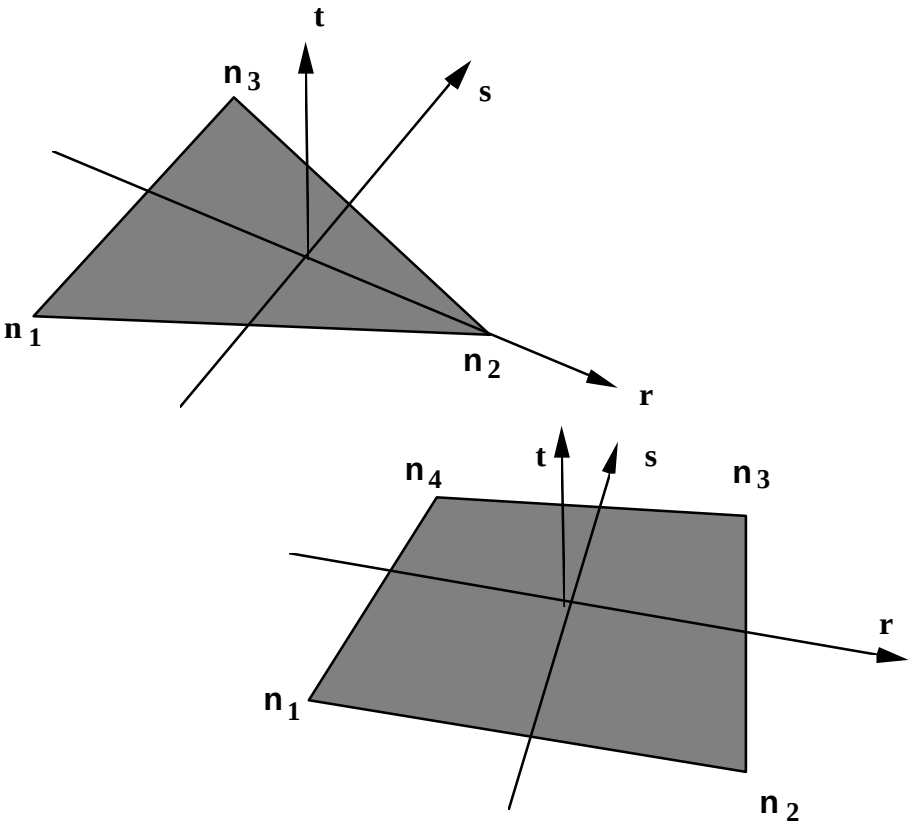


Figure 18.3. Nodal numbering for pressure cards. Positive pressure acts in the negative t-direction.

LOAD_SHELL_*OPTION

Options include:

ELEMENT
SET

Purpose: Apply the distributed pressure load over one shell element or shell element set. The numbering of the shell nodal connectivities must follow the right hand rule with positive pressure acting in the negative t-direction. See Figure 18.3.

Card Format

	1	2	3	4	5	6	7	8
Variable	EID/ESID	LCID	SF	AT				
Type	I	I	F	F				
Default	none	none	1.	0.				
Remarks	1	1	2					

VARIABLE	DESCRIPTION
EID/ESID	Shell ID (SID) or shell set ID (SSID), see *ELEMENT_SHELL or *SET_SHELL.
LCID	Load curve ID, see *DEFINE_CURVE.
SF	Load curve scale factor
AT	Arrival time for pressure or birth time of pressure.

Remarks:

1 If LCID is input as -1, then the Brode function is used to determine the pressure for the segments, see also *LOAD_BRODE.

2. The load curve multipliers may be used to increase or decrease the pressure. The time value is not scaled.

***LOAD_SUPERPLASTIC_FORMING**

Purpose: Perform superplastic forming (SPF) analyses. This option can be applied to both solid and shell elements. The pressure loading controlled by the load curve ID given below is scaled to maintain a constant maximum strain rate.

This option must be used with material model 64, *MAT_RATE_SENSITIVE_POWERLAW_PLASTICITY, for strain rate sensitive, powerlaw plasticity. For the output of data, see *DATABASE_SUPERPLASTIC_FORMING. Mass scaling is recommended in SPF applications.

Card Format

	1	2	3	4	5	6	7	8
Variable	LCP1	CSP1	NCP1	LCP2	CSP2	NCP2		
Type	I	I	F	I	I	F		
Default	none	none	none.	none	none	none		
Remarks				1	1	1		

	1	2	3	4	5	6	7	8
Variable	ERATE	SCMIN	SCMAX	NCYL				
Type	F	F	F	I				
Default	none	none	none.	0				
Remarks				2				

VARIABLE	DESCRIPTION
LCP1	Load curve number for Phase I pressure loading, see *DEFINE_CURVE.
CSP1	Contact surface number to determine completion of Phase 1.
NCP1	Percent of nodes in contact to terminate Phase I, see *CONTACT_ OPTION.
LCP2	Load curve number for Phase II pressure loading (reverse), see *DEFINE_ CURVE.
CSP2	Contact surface to determine completion of Phase II, see *CONTACT_ OPTION.
NCP2	Percent of nodes in contact to terminate Phase II.
ERATE	Desired strain rate. This is the time derivative of the logarithmic strain.
SCMIN	Minimum allowable value for load curve scale factor. To maintain a constant strain rate the pressure curve is scaled. In the case of a snap through buckling the pressure may be removed completely. By putting a value here the pressure will continue to act but at a value giveN by this scale factor multiplYing the pressure curve.
SCMAX	Maximum allowable value for load curve scale factor. Generally, it is a good idea to put a value here to keep the pressure from going to unreasonable values after full contact has been attained. When full contact is achieved the strain rates will approach zero and pressure will go to infinity unless it is limited or the calculation terminates.
NCYL	Number of cycles for monotonic pressure after reversal.

Remarks:

1. Optionally, a second phase can be defined. In this second phase a unique set of pressure segments must be defined whose pressure is controlled by load curve 2. During the first phase, the pressure segments of load curve 2 are inactive, and, likewise, during the second phase the pressure segments of the first phase are inactive. When shell elements are used the complete set of pressure segments can be repeated in the input with a sign reversal used on the load curve. When solid elements are used the pressure segments for each phase will, in general, be unique.

2. This is an ad hoc parameter which should probably not be used.

3. The output files named: “pressure”, “curve1”, and “curve2”, may be ploted by LS-TAURUS in PHS3 using the SUPERPL command. The file “curve2” is created only if the second phase is active. See *DATABASE_SUPERPLASTIC_FORMING.

4. The constraint method contact, *CONTACT_CONSTRAINT_NODES_TO_SURFACE, is recommended for superplastic forming simulations since the penalty methods are not as reliable when mass scaling is applied. Generally, in superplastic simulations mass scaling is used to enable the calculation to be carried out in real time.

LOAD_THERMAL_*OPTION

Options include:

CONSTANT
CONSTANT_NODE
LOAD_CURVE
TOPAZ
VARIABLE
VARIABLE_NODE

Purpose: To define nodal temperatures that thermally load the structure. Nodal temperatures defined by the *LOAD_THERMAL_OPTION method are all applied in a structural only analysis. They are ignored in a thermal only or coupled thermal/structural analysis, see *CONTROL_THERMAL_ OPTION.

All the *LOAD_THERMAL options cannot be used in conjunction with each other. Only those of the same thermal load type, as defined below in column 2, may be used together.

*LOAD_THERMAL_CONSTANT	- Thermal load type 1
*LOAD_THERMAL_CONSTANT_NODE	- Thermal load type 1
*LOAD_THERMAL_LOAD_CURVE	- Thermal load type 2
*LOAD_THERMAL_TOPAZ	- Thermal load type 3
*LOAD_THERMAL_VARIABLE	- Thermal load type 4
*LOAD_THERMAL_VARIABLE_NODE	- Thermal load type 4

*LOAD_THERMAL_CONSTANT

Purpose: Define nodal sets giving the temperature that remains constant for the duration of the calculation. The reference temperature state is assumed to be a null state with this option. A nodal temperature state, read in above and held constant throughout the analysis, dynamically loads the structure. Thus, the temperature defined can also be seen as a relative temperature to a surrounding or initial temperature.

Card Format

Card 1

Variable	NSID	NSIDEX	BOXID					
Type	I	I	I					
Default	none	0.	0.					

Card 2

Variable	T	TE						
Type	F	F						
Default	0.	0.						

VARIABLE	DESCRIPTION
NSID	Nodal set ID containing nodes for initial temperature (see *SET_NODES): EQ.0: all nodes are included:
NSIDEX	Nodal set ID containing nodes that are exempted from the imposed temperature (optional).
BOXID	All nodes in box which belong to NSID are initialized. Others are excluded (optional).
T	Temperature
TE	Temperature of exempted nodes (optional)

***LOAD_THERMAL_CONSTANT_NODE**

Purpose: Define nodal temperature that remains constant for the duration of the calculation. The reference temperature state is assumed to be a null state with this option. A nodal temperature state, read in above and held constant throughout the analysis, dynamically loads the structure. Thus, the temperature defined can also be seen as a relative temperature to a surrounding or initial temperature.

Card Format

	1	2	3	4	5	6	7	8
Variable	NID	T						
Type	I	F						
Default	none	0.						

VARIABLE	DESCRIPTION
NID	Node ID
T	Temperature

*LOAD_THERMAL_LOAD_CURVE

Purpose: Nodal temperatures will be uniform throughout the model and will vary according to a load curve. It is assumed that the temperatures refer to a null state at the beginning and are thus relative temperatures. They dynamically load the structure.

Card Format

	1	2	3	4	5	6	7	8
Variable	LCID							
Type	I							
Default	none							

VARIABLE	DESCRIPTION
LCID	Load curve ID, see *DEFINE_CURVE to define temperature versus time.

***LOAD_THERMAL_TOPAZ**

Purpose: Nodal temperatures will be read in from the TOPAZ3D database. This file is defined in the EXECUTION SYNTAX, see INTRODUCTION.

*LOAD_THERMAL_VARIABLE

Purpose: Define nodal sets giving the temperature that is variable in the duration of the calculation. The reference temperature state is assumed to be a null state with this option. A nodal temperature state, read in above and varied according to the load curve, dynamically loads the structure. Thus, the defined temperatures are relative temperatures to an initial reference temperature.

Card Format

Card 1

Variable	NSID	NSIDEX	BOXID					
Type	I	I	I					
Default	none	0.	0.					

Card 2

Variable	TS	TB	LCID	TSE	TBE	LCIDE		
Type	F	F	I	F	F	I		
Default	0.	0.	none	0.	0.	none		
Remark	1	1	1	1	1			

VARIABLE	DESCRIPTION
NSID	Nodal set ID containing nodes (see *SET_NODE_OPTION): EQ.0: all nodes are included.
NSIDEX	Nodal set ID containing nodes that are exempted (optional), see *SET_NODE_OPTION.
BOXID	All nodes in box which belong to NSID are initialized. Others are excluded.
TS	Scaled temperature.

VARIABLE	DESCRIPTION
TB	Base temperature.
LCID	Load curve ID that multiplies the scaled temperature, see *DEFINE_CURVE.
TSE	Scaled temperature of the exempted nodes (optional).
TBE	Base temperature of the exempted nodes (optional).
LCIDE	Load curve ID that multiplies the scaled temperature of the exempted nodes (optional), see *DEFINE_CURVE.

Remark:

1. The temperature is defined as

$$T = T_{base} + T_{scale}\, f(t)$$

where

$f(t)$ is the current value of the load curve,

T_{scale} is the scaled temperature,

T_{base} is the base temperature.

*LOAD_THERMAL_VARIABLE_NODE

Purpose: Define nodal temperature that are variable during the calculation. The reference temperature state is assumed to be a null state with this option. A nodal temperature state read in and varied according to the load curve dynamically loads the structure. Thus, the defined temperatures are relative temperatures to an initial reference temperature.

Card Format

	1	2	3	4	5	6	7	8
Variable	NID	TS	TB	LCID				
Type	I	F	F	I				
Default	none	0.	0.	none				

VARIABLE	DESCRIPTION
NID	Node ID
TS	Scaled temperature
TB	Base temperature
LCID	Load curve ID that multiplies the scaled temperature, see *DEFINE_CURVE.

The temperature is defined as

$$T = T_{base} + T_{scale}\, f(t)$$

where

f(t) is the current value of the loadcurve

T_{scale} is the scaled temperature

T_{base} is the base temperature

*MAT

LS-DYNA3D has historically referenced materials by type identifiers. Below these identifiers are given with the corresponding keyword name. The numbers in brackets identify the element formulations for which the material model is implemented:

0 - Solids,
1H - Hughes-Liu beam,
1B - Belytschko resultant beam,
1I - Belytschko integrated solid and tubular beams,
1T - Truss,
1D - Discrete beam,
2 - Shells,
3 - Thick shells.
4 - Special airbag element.

TYPE 1:*MAT_ELASTIC [0,1H, 1B, 1I, 1T,2,3]

TYPE 2:*MAT_ORTHOTROPIC_ELASTIC [0,2,3]

TYPE 3:*MAT_PLASTIC_KINEMATIC [0,1H, 1I, 1T,2,3]

TYPE 4:*MAT_ELASTIC_PLASTIC_THERMAL [0,2,3]

TYPE 5:*MAT_SOIL_AND_FOAM [0]

TYPE 6:*MAT_VISCOELASTIC [0,1H]

TYPE 7:*MAT_BLATZ-KO_RUBBER [0,2]

TYPE 8:*MAT_HIGH_EXPLOSIVE_BURN [0]

TYPE 9:*MAT_NULL [0]

TYPE 10:*MAT_ELASTIC_PLASTIC_HYDRO [0]

TYPE 11:*MAT_STEINBERG [0]

TYPE 12:*MAT_ISOTROPIC_ELASTIC_PLASTIC [0,2,3]

TYPE 13:*MAT_ISOTROPIC_ELASTIC_FAILURE [0]

TYPE 14:*MAT_SOIL_AND_FOAM_FAILURE [0]

TYPE 15:*MAT_JOHNSON_COOK [0,2]

TYPE 16:*MAT_PSEUDO_TENSOR [0]

TYPE 17:*MAT_ORIENTED_CRACK [0]

TYPE 18:*MAT_POWER_LAW_PLASTICITY [0,1H,2,3]

TYPE 19:*MAT_STRAIN_RATE_DEPENDENT_PLASTICITY [0,2,3]

TYPE 20:*MAT_RIGID [0,1H,1B,1T,2,3]

TYPE 21:*MAT_ORTHOTROPIC_THERMAL [0,2,3]

TYPE 22:*MAT_COMPOSITE_DAMAGE [0,2,3]

TYPE 23:*MAT_TEMPERATURE_DEPENDENT_ORTHOTROPIC [0,2,3]

TYPE 24:*MAT_PIECEWISE_LINEAR_PLASTICITY [0,1H,2,3]

TYPE 25:*MAT_GEOLOGIC_CAP_MODEL[0]

TYPE 26:*MAT_HONEYCOMB [0]

TYPE 27:*MAT_MOONEY-RIVLIN_RUBBER [0,2]

TYPE 28:*MAT_RESULTANT_PLASTICITY [1B,2]

TYPE 29:*MAT_FORCE_LIMITED [1B]

TYPE 30:*MAT_CLOSED_FORM_SHELL_PLASTICITY [2,3]

TYPE 31:*MAT_FRAZER-NASH_RUBBER [0]

TYPE 32:*MAT_LAMINATED_GLASS [2,3]

TYPE 33:*MAT_BARLAT_ANISOTROPIC_PLASTICITY [0,2,3]

TYPE 34:*MAT_FABRIC [4]

TYPE 35:*MAT_PLASTIC_GREEN-NAGHDI_RATE [0]

TYPE 36:*MAT_3-PARAMETER_BARLAT [2]

TYPE 37:*MAT_TRANSVERSELY_ANISOTROPIC_ELASTIC_PLASTIC [2,3]

TYPE 38:*MAT_BLATZ-KO_FOAM [0,2]

TYPE 39:*MAT_FLD_TRANSVERSELY_ANISOTROPIC [2,3]

TYPE 40:*MAT_NONLINEAR_ORTHOTROPIC [2]

TYPE 41-50:*MAT_USER_DEFINED_MATERIALS

TYPE 51:*MAT_BAMMAN [0,2,3]

TYPE 52:*MAT_BAMMAN_DAMAGE [0,2,3]

TYPE 53:*MAT_CLOSED_CELL_FOAM [0]

TYPE 54:*MAT_ENHANCED_COMPOSITE_DAMAGE [2]

TYPE 57:*MAT_LOW_DENSITY_FOAM [0]

TYPE 59:*MAT_COMPOSITE_FAILURE_MODEL [0,2]

TYPE 60:*MAT_ELASTIC_WITH_VISCOSITY [0,2]

TYPE 61:*MAT_KELVIN-MAXWELL_VISCOELASTIC [0]

TYPE 62:*MAT_VISCOUS_FOAM [0]

TYPE 63:*MAT_CRUSHABLE_FOAM [0]

TYPE 64:*MAT_RATE_SENSITIVE_POWERLAW_PLASTICITY [0,2,3]

TYPE 65:*MAT_MODIFIED_ZERILLI-ARMSTRONG [0]

TYPE 66:*MAT_LINEAR_ELASTIC_DISCRETE_BEAM [1D]

TYPE 67:*MAT_NONLINEAR_ELASTIC_DISCRETE_BEAM [1D]

TYPE 68:*MAT_NONLINEAR_PLASTIC_DISCRETE_BEAM [1D]

TYPE 69:*MAT_SID_DAMPER_DISCRETE_BEAM [1D]

TYPE 70:*MAT_HYDRAULIC_GAS_DAMPER_DISCRETE_BEAM [1D]

TYPE 71:*MAT_CABLE_DISCRETE_BEAM [1D]

TYPE 75:*MAT_BILKHU/DUBOIS_FOAM [0]

TYPE 76:*MAT_GENERAL_VISCOELASTIC [0]

TYPE 77:*MAT_HYPERELASTIC_RUBBER [0] and *MAT_OGDEN_RUBBER [0]

TYPE 78:*MAT_SOIL_CONCRETE [0]

TYPE 79:*MAT_HYSTERETIC_SOIL [0]

TYPE 81:*MAT_PLASTICITY_WITH_DAMAGE [2]

TYPE 86:*MAT_ORTHOTROPIC_VISCOELASTIC [2]

TYPE 87:*MAT_CELLULAR_RUBBER [0]

TYPE 90:*MAT_ACOUSTIC [0]

For the discrete springs and dampers eight materials are available:

*MAT_SPRING_ELASTIC

*MAT_DAMPER_VISCOUS

*MAT_SPRING_ELASTOPLASTIC

*MAT_SPRING_NONLINEAR_ELASTIC

*MAT_DAMPER_NONLINEAR_VISCOUS

*MAT_SPRING_GENERAL_NONLINEAR

*MAT_SPRING_MAXWELL

*MAT_SPRING_INELASTIC

For the seatbelts one material is available:

*MAT_SEATBELT

For thermal materials in a coupled structural/thermal or thermal only analysis six materials are available. These materials are related to the structural material via the *PART card. Thermal materials are defined only for solid and shell elements.

*MAT_THERMAL_ISOTROPIC

*MAT_THERMAL_ORTHOTROPIC

*MAT_THERMAL_ISOTROPIC_TD

*MAT_THERMAL_ORTHOTROPIC_TD

*MAT_THERMAL_ISOTROPIC_PHASE_CHANGE

*MAT_THERMAL_ISOTROPIC_TD_LC

***MAT_ELASTIC_OPTION**

This is Material Type 1. This is an isotropic elastic material and is available for beam, shell, and solid elements in LS-DYNA3D. A specialization of this material allows the modeling of fluids.

Options include:

<BLANK>
FLUID

such that the keyword cards appear:

***MAT_ELASTIC**
***MAT_ELASTIC_FLUID**

The fluid option is valid for solid elements only.

Define the following card for all options:

Card Format

	1	2	3	4	5	6	7	8
Variable	MID	RO	E	PR	DA	DB	K	
Type	I	F	F	F	F	F	F	
Default	none	none	none	none	0.0	0.0	0.0	

Define the following extra card for the FLUID option:

Card Format

	1	2	3	4	5	6	7	8
Variable	VC	CP						
Type	F	F						
Default	none	1.0E+20						

VARIABLE	DESCRIPTION
MID	Material identification. A unique number has to be chosen.
RO	Mass density.
E	Young's modulus.
PR	Poisson's ratio.
DA	Axial damping factor (used for Belytschko-Schwer beam, type 1, only).
DB	Bending damping factor (used for Belytschko-Schwer beam, type 1, only).
K	Bulk Modulus (define for fluid option only).
VC	Tensor viscosity coefficient, values between .1 and .5 should be okay.
CP	Cavitation pressure (default = 1.0e+20).

The axial and bending damping factors are used to damp down numerical noise. The update of the force resultants, F_i , and moment resultants, M_i , includes the damping factors:

$$F_i^{n+1} = F_i^n + \left(1 + \frac{DA}{\Delta t}\right)\Delta F_i^{n+\frac{1}{2}}$$

$$M_i^{n+1} = M_i^n + \left(1 + \frac{DB}{\Delta t}\right)\Delta M_i^{n+\frac{1}{2}}$$

For the fluid option the bulk modulus (K) has to be defined as Young's modulus, and Poission's ratio are ignored. With the fluid option fluid-like behavior is obtained where the bulk modulus, K, and pressure rate, $\dot{p}$, are given by:

$$K = \frac{E}{3(1-2\nu)}$$

$$\dot{p} = -K\ \dot{\varepsilon}_{ii}$$

and the shear modulus is set to zero. A tensor viscosity is used which acts only the deviatoric stresses, S_{ij}^{n+1}, given in terms of the damping coefficient as:

$$S_{ij}^{n+1} = VC \cdot \Delta L \cdot a \cdot \rho \dot{\varepsilon}_{ij}^{'}$$

where ΔL, is a characteristic element length, a is the fluid bulk sound speed, ρ is the fluid density, and $\dot{\varepsilon}_{ij}^{'}$ is the deviatoric strain rate.

***MAT_OPTION TROPIC_ELASTIC**

This is Material Type 2. This material is valid for modeling the elastic-orthotropic behavior of solids, shells, and thick shells. An anisotropic option is available for solid elements.

Options include:

ORTHO
ANISO

such that the keyword cards appear:

***MAT_ORTHOTROPIC_ELASTIC** (4 cards follow)
***MAT_ANISOTROPIC_ELASTIC** (5 cards follow)

Card Format of Cards 1 and 2 for the ORTHO option.

Card 1

Variable	MID	RO	EA	EB	EC	PRBA	PRCA	PRCB
Type	I	F	F	F	F	F	F	F

Card 2

Variable	GAB	GBC	GCA	AOPT				
Type	F	F	F	F				

Card Format of Cards 1, 2, and 3 for the ANISO option.

Card 1

1	2	3	4	5	6	7	8
Variable	MID	RO	C11	C12	C22	C13	C23	C33
Type	I	F	F	F	F	F	F	F

Card 2

Variable	C14	C24	C34	C44	C15	C25	C35	C45
Type	F	F	F	F	F	F	F	F

Card 3

Variable	C55	C16	C26	C36	C46	C56	C66	AOPT
Type	F	F	F	F	F	F	F	F

Card Format of Cards 3/4 and 4/5 for the ORTHO/ANISO options.

Card 3/4

Variable	XP	YP	ZP	A1	A2	A3		
Type	F	F	F	F	F	F		

Card 4/5

Variable	V1	V2	V3	D1	D2	D3	BETA	
Type	F	F	F	F	F	F	F	

VARIABLE	DESCRIPTION
MID	Material identification. A unique number has to be chosen.
RO	Mass density.

Define for the ORTHO option only:

EA	E_a, Young's modulus in a-direction.
EB	E_b, Young's modulus in b-direction.
EC	E_c, Young's modulus in c-direction.
PRBA	ν_{ba}, Poisson's ratio ba.
PRCA	ν_{ca}, Poisson's ratio ca.
PRCB	ν_{cb}, Poisson's ratio cb.
GAB	G_{ab}, shear modulus ab.
GBC	G_{bc}, shear modulus bc.
GCA	G_{ca}, shear modulus ca.

Due to symmetry define the upper triangular Cij's for the ANISO option only:

C11	The 1,1 term in the 6 ∞ 6 anisotropic constitutive matrix. Note that 1 corresponds to the *a* material direction
C12	The 1,2 term in the 6 ∞ 6 anisotropic constitutive matrix. Note that 2 corresponds to the *b* material direction
.	.
.	.
.	.
C66	The 6,6 term in the 6 ∞ 6 anisotropic constitutive matrix.

Define for both options:

AOPT	Material axes option, see Figure 19.1:

VARIABLE	DESCRIPTION
	EQ. 0.0: locally orthotropic with material axes determined by element nodes as shown in Figure 19.1. Nodes 1, 2, and 4 of an element are identical to the Nodes used for the definition of a coordinate system as by *DEFINE_COORDINATE_NODES. EQ. 1.0: locally orthotropic with material axes determined by a point in space and the global location of the element center; this is the a-direction. EQ. 2.0: globally orthotropic with material axes determined by vectors defined below, as with *DEFINE_COORDINATE_VECTOR. EQ. 3.0: applicable to shell elements only. This option determines locally orthotropic material axes by offsetting the material axes by an angle to be specified from a line in the plane of the shell determined by taking the cross product of the vector **v** defined below with the shell normal vector. EQ. 4.0: locally orthotropic in cylindrical coordinate system with the material axes determined by a vector **v** and an orginating point, P. They define the axis of symmetry.
XP,YP,ZP	x_p y_p z_p, define coordinates of point **p** for AOPT = 1 and 4.
A1,A2,A3	a_1 a_2 a_3, define components of vector **a** for AOPT = 2.
D1,D2,D3	d_1 d_2 d_3, define components of vector **d** for AOPT = 2.
V1,V2,V3	v_1 v_2 v_3, define components of vector **v** for AOPT = 3 and 4.
PSI	Material angle for AOPT = 3, which may be overridden on the element card, see *ELEMENT_SHELL.

The material law that relates stresses to strains is defined as:

$$\underset{\sim}{C} = \underset{\sim}{T}^T \underset{\sim L}{C} \underset{\sim}{T}$$

where $\underset{\sim}{T}$ is a transformation matrix, and $\underset{\sim L}{C}$ is the constitutive matrix defined in terms of the material constants of the orthogonal material axes, a, b, and c. The inverse of $\underset{\sim L}{C}$ is defined as

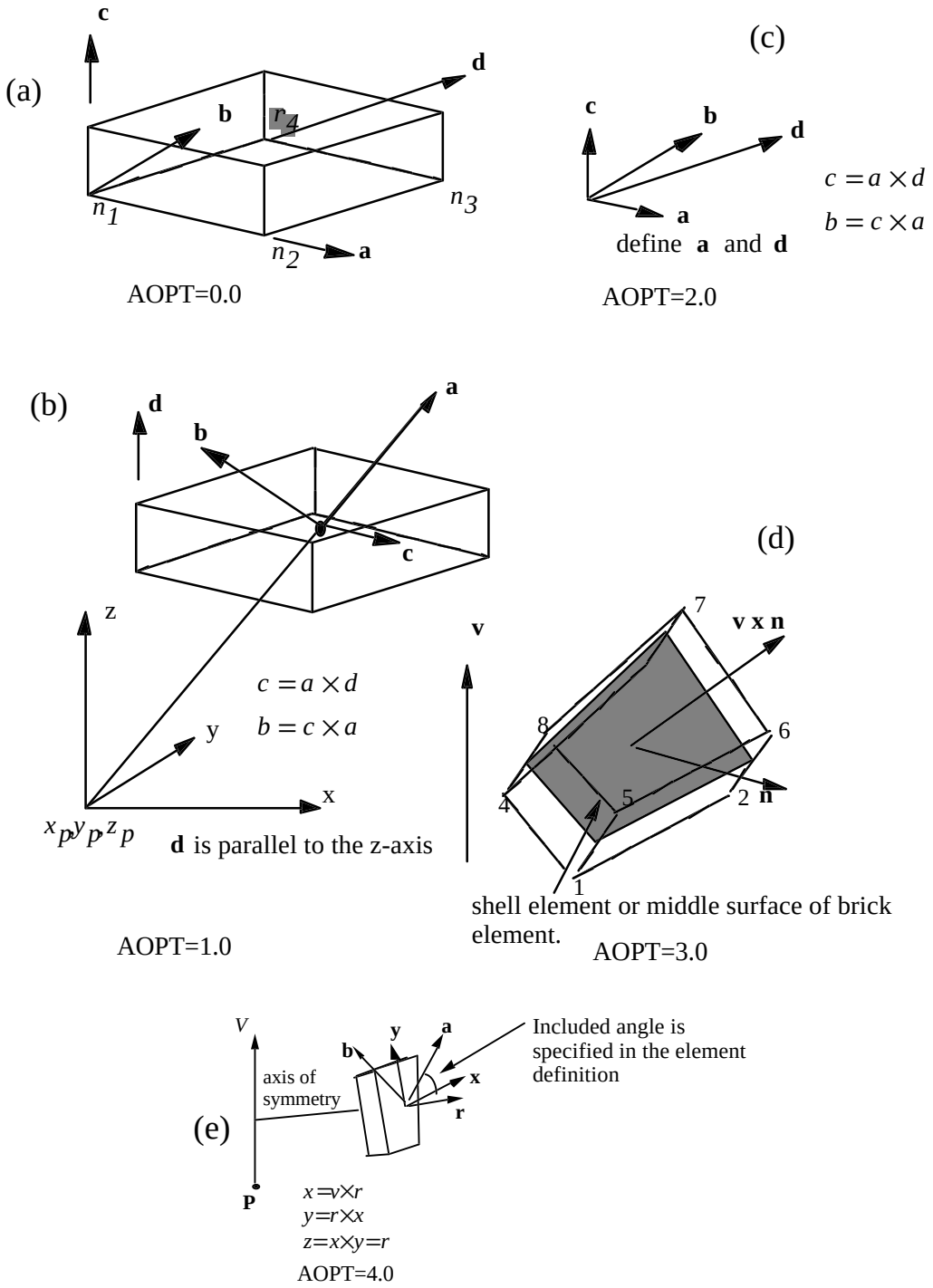


Figure 19.1. Options for determining principal material axes: (a) AOPT = 0.0, (b) AOPT = 1.0, (c) AOPT = 2.0,. Note that $\underset{\sim}{c} = \underset{\sim}{a} \times \underset{\sim}{d}$ *and that* $\underset{\sim}{b} = \underset{\sim}{c} \times \underset{\sim}{a}$, (d) AOPT = 3.0, and (e) AOPT=4.0 for brick elements.

$$
\underset{\sim L}{C^{-1}} = \begin{bmatrix}
\frac{1}{E_a} & -\frac{\upsilon_{ba}}{E_b} & -\frac{\upsilon_{ca}}{E_c} & 0 & 0 & 0 \\
-\frac{\upsilon_{ab}}{E_a} & \frac{1}{E_b} & -\frac{\upsilon_{cb}}{E_c} & 0 & 0 & 0 \\
-\frac{\upsilon_{ac}}{E_a} & -\frac{\upsilon_{bc}}{E_b} & \frac{1}{E_c} & 0 & 0 & 0 \\
0 & 0 & 0 & \frac{1}{G_{ab}} & 0 & 0 \\
0 & 0 & 0 & 0 & \frac{1}{G_{bc}} & 0 \\
0 & 0 & 0 & 0 & 0 & \frac{1}{G_{ca}}
\end{bmatrix}
$$

Note that $\frac{\upsilon_{ab}}{E_a} = \frac{\upsilon_{ba}}{E_b}$, $\frac{\upsilon_{ca}}{E_c} = \frac{\upsilon_{ac}}{E_a}$, $\frac{\upsilon_{cb}}{E_c} = \frac{\upsilon_{bc}}{E_b}$.

***MAT_PLASTIC_KINEMATIC**

This is Material Type 3. This model is suited to model isotropic and kinematic hardening plasticity with the option of including rate effects. It is a very cost effective model and is available for beam (Hughes-Liu), shell, and solid elements.

Card Format

Card 1

Variable	MID	RO	E	PR	SIGY	ETAN	BETA	
Type	I	F	F	F	F	F	F	
Default	none	none	none	none	none	0.0	0.0	

Card 2

Variable	SRC	SRP	FS					
Type	F	F	F					
Default	not used	not used	not used					

VARIABLE	DESCRIPTION
MID	Material identification. A unique number has to be chosen.
RO	Mass density.
E	Young's modulus.
PR	Poisson's ratio.
SIGY	Yield stress.
ETAN	Plastic tangent hardening modulus, see Figure 19.2.
BETA	Hardening parameter, $0 < \beta' < 1$. See comments below.

VARIABLE	DESCRIPTION
SRC	Strain rate parameter, C, for Cowper Symonds strain rate model, see below. If zero, rate effects are not considered.
SRP	Strain rate parameter, P, for Cowper Symonds strain rate model, see below. If zero, rate effects are not considered.
FS	Failure strain for eroding elements.

Strain rate is accounted for using the Cowper and Symonds model which scales the yield stress with the factor

$$1+\left(\frac{\dot{\varepsilon}}{C}\right)^{1/P}$$

where $\dot{\varepsilon}$ is the strain rate. To ignore strain rate effects set both SRC and SRP to zero.

Kinematic, isotropic, or a combination of kinematic and isotropic hardening may be specified by varying β' between 0 and 1. For β' equal to 0 and 1, respectively, kinematic and isotropic hardening are obtained as shown in Figure 19.2. For isotropic hardening, $\beta' = 1$, Material Model 12, *MAT_ISOTROPIC_ELASTIC_PLASTIC, requires less storage and is more efficient. Whenever possible, Material 12 is recommended for solid elements, but for shell elements it is less accurate and thus material 12 is not recommend in this case.

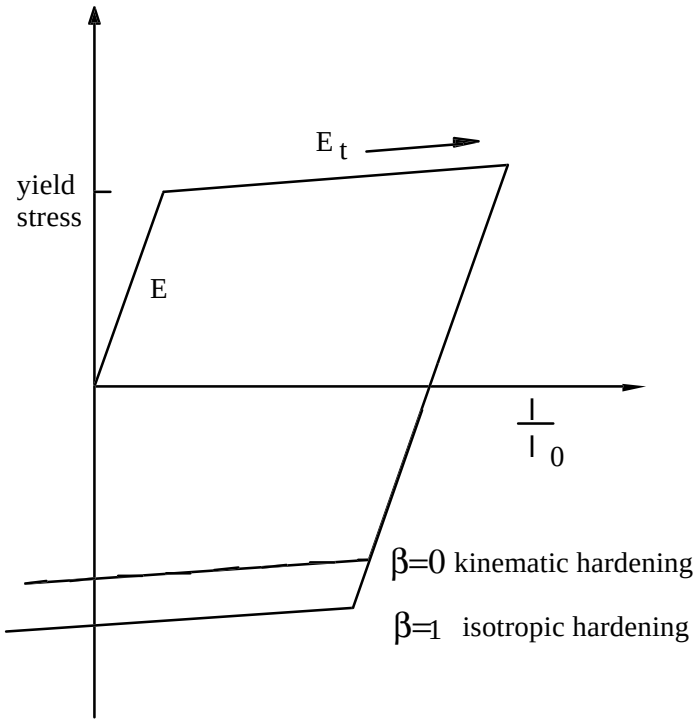


Figure 19.2. Elastic-plastic behavior with kinematic and isotropic hardening where l_0 and l are undeformed and deformed lengths of uniaxial tension specimen. E_t is the slope of the bilinear stress strain curve.

*MAT_ELASTIC_PLASTIC_THERMAL

This is Material Type 4. Temperature dependent material coefficients can be defined. A maximum of eight temperatures with the corresponding data can be defined. A minimum of two points is needed.

Card Format

Card 1

Variable	MID	RO						
Type	I	F						

Card 2

Variable	T1	T2	T3	T4	T5	T6	T7	T8
Type	F	F	F	F	F	F	F	F

Card 3

Variable	E1	E2	E3	E4	E5	E6	E7	E8
Type	F	F	F	F	F	F	F	F

Card 4

Variable	PR1	PR2	PR3	PR4	PR5	PR6	PR7	PR8
Type	F	F	F	F	F	F	F	F

Card Format (no defaults are assumed)

Card 5

Variable	ALPHA1	ALPHA2	ALPHA3	ALPHA4	ALPHA5	ALPHA6	ALPHA7	ALPHA8
Type	F	F	F	F	F	F	F	F

Card 6	1	2	3	4	5	6	7	8

Variable	SIGY1	SIGY2	SIGY3	SIGY4	SIGY5	SIGY6	SIGY7	SIGY8
Type	F	F	F	F	F	F	F	F

Card 7

Variable	ETAN1	ETAN2	ETAN3	ETAN4	ETAN5	ETAN6	ETAN7	ETAN8
Type	F	F	F	F	F	F	F	F

VARIABLE	DESCRIPTION
MID	Material identification. A unique number must be chosen.
RO	Mass density.
TI	Temperatures. The minimum is 2, the maximum is 8.
EI	Corresponding Young's moduli at temperature TI.
PRI	Corresponding Poisson's ratios.
ALPHAI	Corresponding coefficients of thermal expansion.
SIGYI	Corresponding yield stresses.
ETANI	Corresponding plastic hardening moduli.

At least two temperatures and their corresponding material properties must be defined. The analysis will be terminated if a material temperature falls outside the range defined in the input. If a thermoelastic material is considered, do not define SIGY and ETAN. The coefficient of thermal expansion is defined with respect to the reference temperature at the beginning of the calculation for the material. Thus, the thermal strain rate becomes:

$$\dot{\varepsilon}_{ij}^{T} = \left[\frac{d\alpha}{dT}\left(T - T_{ref}\right) + \alpha \right] \dot{T} \delta_{ij}$$

***MAT_SOIL_AND_FOAM**

This is Material Type 5. This is a very simple model and works in some ways like a fluid. It should be used only in situations when soils and foams are confined within a structure or when geometric boundaries are present.

Card Format

Card 1

Variable	MID	RO	G	BULK	A0	A1	A2	PC
Type	I	F	F	F	F	F	F	F

Card 2

Variable	VCR							
Type	F							

Card 3

Variable	EPS1	EPS2	EPS3	EPS4	EPS5	EPS6	EPS7	EPS8
Type	F	F	F	F	F	F	F	F

Card 4

Variable	EPS9	EPS10						
Type	F	F						

Card 5

Variable	P1	P2	P3	P4	P5	P6	P7	P8
Type	F	F	F	F	F	F	F	F

Card 6

Variable	P9	P10						
Type	F	F						

VARIABLE	DESCRIPTION
MID	Material identification. A unique number has to be chosen.
RO	Mass density.
G	Shear modulus.
K	Bulk modulus for unloading used for VCR=0.0.
A0	Yield function constant for plastic yield function below.
A1	Yield function constant for plastic yield function below.
A2	Yield function constant for plastic yield function below.
PC	Pressure cutoff for tensile fracture.
VCR	Volumetric crushing option: EQ.0.0: on, EQ.1.0: loading and unloading paths are the same.
EPS1,.....	Volumetric strain values (natural logarithmic values), see comments below. A maximum of 10 values are allowed and a minimum of 2 values are necessary. The tabulated values must competely cover the expected values in the analysis. If the first value is not for a volumetric strain value of zero then the point (0.0,0.0) will be automatically generated and upto a further nine additional values may be defined.
P1, P2,..PN	Pressures corresponding to volumetric strain values.

Pressure is positive in compression. Volumetric strain is given by the natural log of the relative volume and is negative in compression. Relative volume is ratio of the current volume to the initial volume at the start of the calculation. The tabulated data should be given in order of increasing compression. If the pressure drops below the cutoff value specified, it is reset to that value. For a detailed description we refer to Kreig [1972].

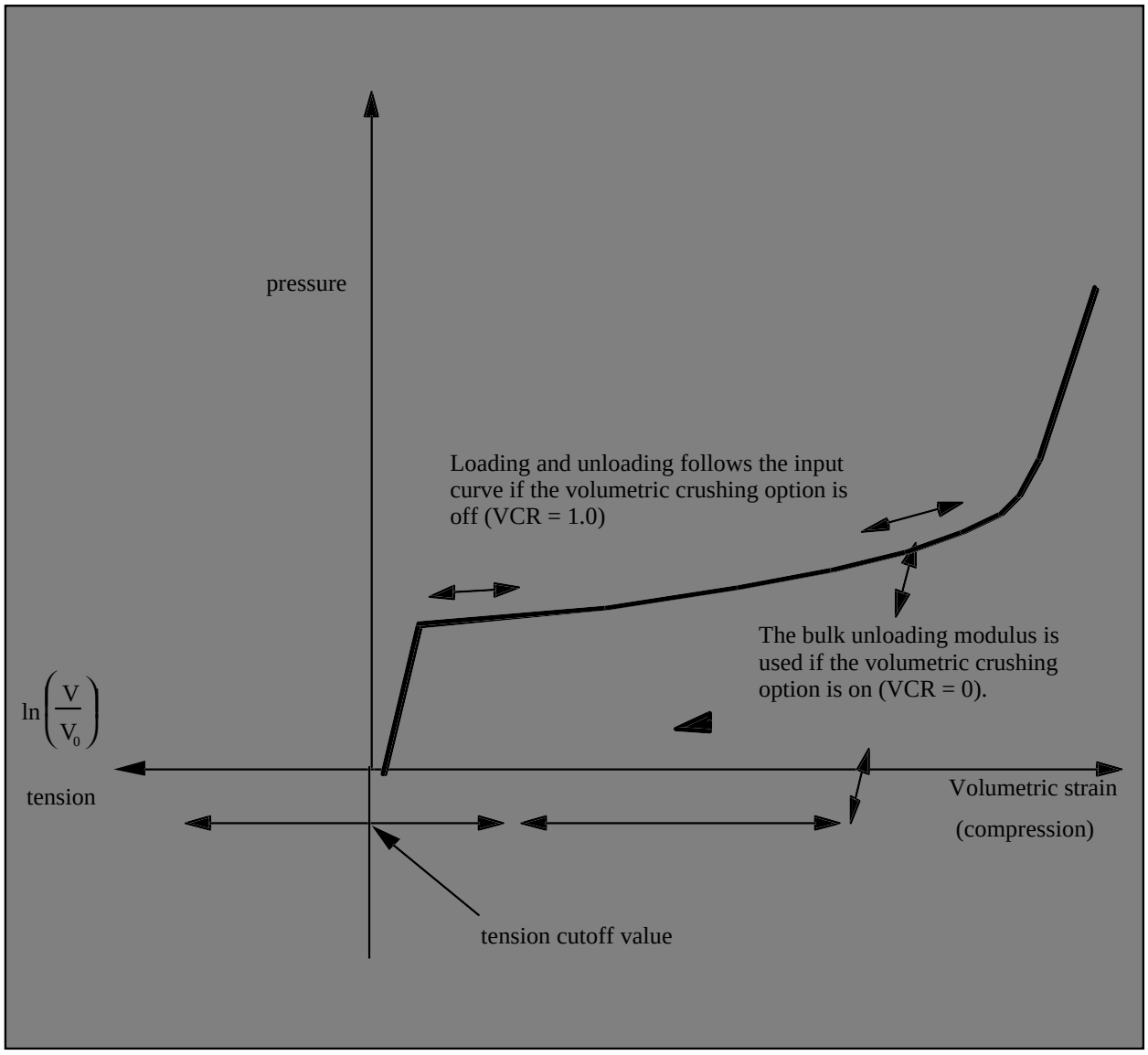

pressure
Loading and unloading follows the input curve if the volumetric crushing option is off (VCR = 1.0)
The bulk unloading modulus is used if the volumetric crushing option is on (VCR = 0).
ln(V/V0)
tension
Volumetric strain
(compression)
tension cutoff value

Figure 19.3. Pressure versus volumetric strain curve for soil and crushable foam model. The volumetric strain is given by the natural logarithm of the relative volume, V.

The deviatoric perfectly plastic yield function, ϕ, is described in terms of the second invariant J_2,

$$J_2 = \frac{1}{2} s_{ij} s_{ij} \ ,$$

pressure, p, and constants a_0, a_1, and a_2 as:

$$\phi = J_2 - \left[a_0 + a_1 p + a_2 p^2 \right] \ .$$

On the yield surface $J_2 = \frac{1}{3}\sigma_y^2$ where σ_y is the uniaxial yield stress, i.e.,

$$\sigma_y = \left[3\left(a_0 + a_1 p + a_2 p^2 \right) \right]^{1/2}$$

There is no strain hardening on this surface. For no pressure hardening, $a_1 = a_2 = 0$, and $(3a_0)^{1/2}$ defines the yield strength.

***MAT_VISCOELASTIC**

This is Material Type 6. This model allows the modeling of viscoelastic behavior for beams (Hughes-Liu), shells, and solids. Also see *MAT_GENERAL_VISCOELASTIC for a more general formulation.

Card Format

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	BULK	G0	GI	BETA		
Type	I	F	F	F	F	F		

VARIABLE	DESCRIPTION
MID	Material identification. A unique number has to be chosen.
RO	Mass density
BULK	Elastic bulk modulus.
G0	Short-time shear modulus, see equations below.
GI	Long-time (infinite) shear modulus, G_∞.
BETA	Decay constant.

The shear relaxation behavior is described by [Hermann and Peterson, 1968]:

$$G(t) = G_\infty + (G_0 - G_\infty)\, e^{-\beta t}$$

A Jaumann rate formulation is used

$$\overset{\nabla}{\sigma}'_{ij} = 2 \int_0^t G(t-\tau)\, D'_{ij}(\tau) dt$$

where the prime denotes the deviatoric part of the stress rate, $\overset{\nabla}{\sigma}_{ij}$, and the strain rate D_{ij}.

*MAT_BLATZ-KO_RUBBER

This is Material Type 7. This one parameter material allows the modeling of nearly incompressible continuum rubber. The Poisson's ratio is fixed to 0.463.

Card Format

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	G					
Type	I	F	F					

VARIABLE	DESCRIPTION
MID	Material identification. A unique number has to be chosen.
RO	Mass density.
G	Shear modulus.

The second Piola-Kirchhoff stress is computed as

$$S_{ij} = G\left[\frac{1}{V}C_{ij} - V^{\left(\frac{1}{1-2\upsilon}\right)}\delta_{ij}\right]$$

where V is the relative volume defined as being the ratio of the current volume to the initial volume, C_{ij} is the right Cauchy-Green strain tensor, and ν is Poisson's ratio, which is set to .463 internally. This stress measure is transformed to the Cauchy stress, $\sigma_{ij,}$ according to the relationship

$$\sigma_{ij} = V^{-1} F_{ik} F_{jl} S_{lk}$$

where F_{ij} is the deformation gradient tensor. Also see Blatz and Ko [1962].

***MAT_HIGH_EXPLOSIVE_BURN**

This is Material Type 8. It allows the modeling of the detonation of a high explosive. In addition an equation of state must be defined. See Wilkins [1969] and Giroux [1972].

Card Format

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	D	PCJ	BETA	K	G	SIGY
Type	I	F	F	F	F	F	F	F

VARIABLE	DESCRIPTION
MID	Material identification. A unique number has to be chosen.
RO	Mass density.
D	Detonation velocity.
PCJ	Chapman-Jouget pressure.
BETA	Beta burn flag, BETA (see comments below): EQ.0.0: beta + programmed burn, EQ.1.0: beta burn only, EQ.2.0: programmed burn only.
K	Bulk modulus (BETA=2.0 only).
G	Shear modulus (BETA=2.0 only).
SIGY	σ_y, yield stress (BETA=2.0 only).

If programmed burn is used the explosive model will behave as an elastic perfectly plastic material if the bulk modulus, shear modulus, and yield stress are defined. Otherwise, any compression of the explosive material will cause detonation.

Burn fractions, which multiply the equations of states for high explosives, control the release of chemical energy for simulating detonations. In the initialization phase, a lighting time t_l is computed for each element by dividing the distance from the detonation point to the center of the element by the detonation velocity D. If multiple detonation points are defined, the closest point determines t_l The burn fraction F is taken as the maximum

$$F = \max (F_1 , F_2)$$

where

$$F_1 = \begin{cases} \dfrac{2(t - t_l)D}{3\left(\dfrac{v_e}{A_{e_{\max}}}\right)} & \text{if } t > t_l \\ 0 & \text{if } t \le t_l \end{cases}$$

$$F_2 = \frac{1 - V}{1 - V_{CJ}}$$

where V_{CJ} is the Chapman-Jouguet relative volume and t is current time. If F exceeds 1, it is reset to unity. This calculation of the burn fraction usually requires several time steps for F to reach unity, thereby spreading the burn front over several elements. After reaching unity, F is held constant.

***MAT_NULL**

This is Material Type 9. This material allows equations of state to be considered without computing deviatoric stresses. Optionally, a viscosity can be defined. Also, erosion in tension and compression is possible.

Sometimes it is advantageous to model contact surfaces via shell elements which are not part of the structure, but are necessary to define areas of contact within nodal rigid bodies or between nodal rigid bodies. Beams and shells that use this material type are completely bypassed in the element processing. The Young's modulus and Poisson's ratio are used only for setting the contact interface stiffnesses, and it is recommended that reasonable values be input.

Card Format

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	PC	MU	TEROD	CEROD	YM	PR
Type	I	F	F	F	F	F	F	F
Defaults	none	none	0.0	0.0	0.0	0.0	0.0	0.0

VARIABLE	DESCRIPTION
MID	Material identification. A unique number has to be chosen.
RO	Mass density
PC	Pressure cutoff (≤ 0.0).
MU	Viscosity coefficient μ (optional).
TEROD	Relative volume, $\frac{V}{V_0}$, for erosion in tension. Typically, use values greater than unity. If zero, erosion in tension is inactive.
CEROD	Relative volume, $\frac{V}{V_0}$, for erosion in compression. Typically, use values less than unity. If zero, erosion in compression is inactive.
YM	Young's modulus (used for null beams and shells only)

VARIABLE	DESCRIPTION
PR	Poisson's ratio (used for null beams and shells only)

The null material must be used with an equation of-state. Pressure cutoff is negative in tension. A viscous stress of the form

$$\sigma_{ij} = \mu \dot{\varepsilon}'_{ij}$$

is computed for nonzero μ where $\dot{\varepsilon}'_{ij}$ is the deviatoric strain rate.

*MAT_ELASTIC_PLASTIC_HYDRO

This is Material Type 10. This material allows the modeling of an elastic-plastic hydynamic material.

Card Format

Card 1

Variable	MID	RO	G	SIGY	EH	PC		
Type	I	F	F	F	F	F		
Default	none	none	none	0.0	0.0	-		

Card 2

Variable	EPS1	EPS2	EPS3	EPS4	EPS5	EPS6	EPS7	EPS8
Type	F	F	F	F	F	F	F	F

Card 3

Variable	EPS9	EPS10	EPS11	EPS12	EPS13	EPS14	EPS15	EPS16
Type	F	F	F	F	F	F	F	F

Card 4

Variable	ES1	ES2	ES3	ES4	ES5	ES6	ES7	ES8
Type	F	F	F	F	F	F	F	F

Card 5

Variable	ES9	ES10	ES11	ES12	ES13	ES14	ES15	ES16
Type	F	F	F	F	F	F	F	F

VARIABLE	DESCRIPTION
MID	Material identification. A unique number has to be chosen.
RO	Mass density.
G	Shear modulus.
SIGY	Yield stress, see comment below.
EH	Plastic hardening modulus, see definition below.
PC	Pressure cutoff (≤ 0.0). If zero, a cutoff of -∞ is assumed.
EPS	Effective plastic strain (logrithmic). Define up to 16 values. Care must be taken that the full range of strains expected in the analysis is covered. Linear extrapolation is used if the strain values exceed the maximum input value.
ES	Effective stress. Define up to 16 values.

If ES and EPS are undefined, the yield stress and plastic hardening modulus are taken from SIGY and EH. In this case, the bilinear stress-strain curve shown in Figure 19.2. is obtained with hardening parameter, β, = 1. The yield strength is calculated as

$$\sigma_y = \sigma_0 + E_h \bar{\varepsilon}^p$$

The quantity E_h is the plastic hardening modulus defined in terms of Young's modulus, E, and the tangent modulus, E_t , as follows

$$E_h = \frac{E_t\, E}{E - E_t} \quad .$$

If ES and EPS are specified, a curve like that shown in Figure 19.4 may be defined. Effective stress is defined in terms of the deviatoric stress tensor, s_{ij}, as:

$$\bar{\sigma} = \left(\frac{3}{2} s_{ij} \; s_{ij} \right)^{1/2}$$

and effective plastic strain by:

$$\bar{\varepsilon}^p = \int_0^t \left(\frac{2}{3} D_{ij}^p \; D_{ij}^p \right)^{1/2} dt \, ,$$

where t denotes time and D_{ij}^p is the plastic component of the rate of deformation tensor. In this case the plastic hardening modulus on Card 1 is ignored and the yield stress is given as

$$\sigma_y = f\left(\bar{\varepsilon}^p\right),$$

where the value for $f\left(\bar{\varepsilon}^p\right)$ is found by interpolation from the data curve.

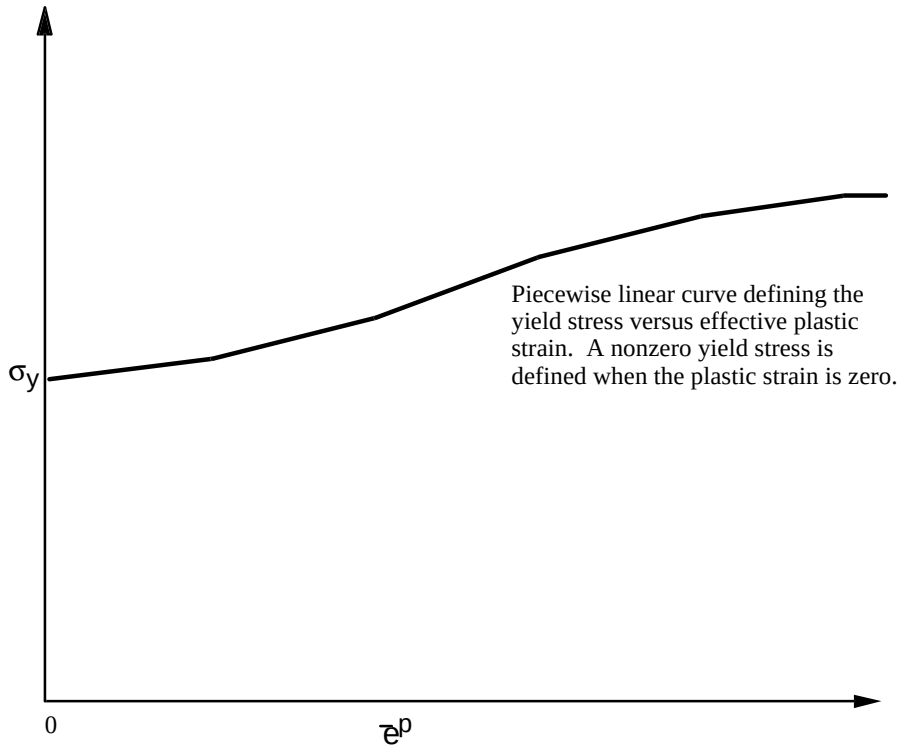

Piecewise linear curve defining the yield stress versus effective plastic strain. A nonzero yield stress is defined when the plastic strain is zero.
σy
0
ε̄p

Figure 19.4. Effective stress versus effective plastic strain curve.

*MAT_STEINBERG

This is Material Type 11. This material is available for modeling materials deforming at very high strain rates (>10^5) and can be used with solid elements. The yield strength is a function of temperature and pressure. An equation of state is determines the pressure.

Card Format

Card 1

Variable	MID	RO	G0	SIGO	BETA	N	GAMA	SIGM
Type	I	F	F	F	F	F	F	F

Card 2

Variable	B	BP	H	F	A	TMO	GAMO	SA
Type	F	F	F	F	F	F	F	F

Card 3

Variable	PC	SPALL	RP	FLAG	MMN	MMX	ECO	EC1
Type	F	F	F	F	F	F	F	F

Card 4

Variable	EC2	EC3	EC4	EC5	EC6	EC7	EC8	EC9
Type	F	F	F	F	F	F	F	F

VARIABLE	DESCRIPTION
MID	Material identification. A unique number has to be chosen.
RO	Mass density.
G0	Basic shear modulus.
SIGO	σ_o, see defining equations.
BETA	β, see defining equations.
N	n, see defining equations.
GAMA	γ_i, initial plastic strain, see defining equations.
SIGM	σ_m, see defining equations.
B	b, see defining equations.
BP	b′, see defining equations.
H	h, see defining equations.
F	f, see defining equations.
A	Atomic weight (if = 0.0, R′ must be defined).
TMO	T_{mo}, see defining equations.
GAMO	γ_o, see defining equations.
SA	a, see defining equations.
PC	p_{min} or $-\sigma_f$
SPALL	Spall type: EQ. 0.0: default set to "2.0", EQ. 1.0: $p \geq p_{min}$, EQ. 2.0: if $\sigma_{max} \geq -p_{min}$ element spalls and tension, $p < 0$, is never allowed, EQ. 3.0: $p < -p_{min}$ element spalls and tension, $p < 0$, is never allowed.
RP	R′. If R′ $\neq$ 0.0, A is not defined.
FLAG	Set to 1.0 for μ coefficients for the cold compression energy fit. Default is η.

VARIABLE	DESCRIPTION
MMN	μ_{min} or η_{min}. Optional μ or η minimum value.
MMX	μ_{max} or η_{max}. Optional μ or η maximum value.
EC0,...EC9	Cold compression energy coefficients (optional).

Users who have an interest in this model are encouraged to study the paper by Steinberg and Guinan which provides the theoretical basis. Another useful reference is the KOVEC user's manual.

In terms of the foregoing input parameters, we define the shear modulus, G, before the material melts as:

$$G = G_0 \left[1 + bpV^{1/3} - h\left(\frac{E_i - E_c}{3R'} - 300 \right) \right] e^{-fE_i / E_m - E_i}$$

where p is the pressure, V is the relative volume, E_c is the cold compression energy:

$$E_c(x) = \int_0^x p\,dx - \frac{900\, R' \exp(ax)}{(1-x)^{2(\gamma_0 - a - 1/2)}} ,$$

$$x = 1 - V,$$

and E_m is the melting energy:

$$E_m(x) = E_c(x) + 3R'T_m(x)$$

which is in terms of the melting temperature $T_m(x)$:

$$T_m(x) = \frac{T_{mo} \exp(2ax)}{V^{2(\gamma_o - a - 1/3)}}$$

and the melting temperature at $\rho = \rho_o$, T_{mo} .

In the above equation R′ is defined by

$$R' = \frac{R\rho}{A}$$

where R is the gas constant and A is the atomic weight. If R′ is not defined, LS-DYNA3D computes it with R in the cm-gram-microsecond system of units.

The yield strength σ_y is given by:

$$\sigma_y = \sigma_0' \left[1 + b' p V^{1/3} - h\left(\frac{E_i - E_c}{3R'} - 300 \right) \right] e^{-fE_i / E_m - E_i}$$

if E_m exceeds E_i. Here, σ_0' is given by:

$$\sigma_y = \sigma_0' \left[1 + \beta\left(\gamma_i + \bar{\varepsilon}^p \right) \right]^n$$

where γ_i is the initial plastic strain. Whenever σ_0' exceeds σ_m, σ_0' is set equal to σ_m. After the materials melts, σ_y and G are set to one half their initial value.

If the coefficients EC0,...,EC9 are not defined above, LS-DYNA3D will fit the cold compression energy to a ten term polynomial expansion either as a function of μ or η depending on the input variable, FLAG, as:

$$E_c\left(\eta^i\right) = \sum_{i=0}^{9} EC_i \eta^i$$

$$E_c\left(\mu^i\right) = \sum_{i=0}^{9} EC_i \mu^i$$

where EC_i is the ith coefficient and:

$$\eta = \frac{\rho}{\rho_o}$$

$$\mu = \frac{\rho}{\rho_o} - 1$$

A linear least squares method is used to perform the fit.

*MAT_ISOTROPIC_ELASTIC_PLASTIC

This is Material Type 12. This is a very low cost isotropic plasticity model for three dimensional solids. For shell elements a simple radial return is used and is not recommended due to lack of accuracy.

Card Format

	1	2	3	4	5	6	7	8
Variable	MID	RO	G	SIGY	ETAN	BULK		
Type	I	F	F	F	F	F		

VARIABLE	DESCRIPTION
MID	Material identification. A unique number has to be chosen.
RO	Mass density.
G	Shear modulus.
SIGY	Yield stress.
ETAN	Plastic hardening modulus.
BULK	Bulk modulus, K.

Here the pressure is integrated in time

$$\dot{p} = -K\ \dot{\varepsilon}_{ii}$$

where $\dot{\varepsilon}_{ii}$ is the volumetric strain rate.

*MAT_ISOTROPIC_ELASTIC_FAILURE

This is Material Type 13. This is a non-iterative plasticity with simple plastic strain failure model.

Card Format

Card 1

Variable	MID	RO	G	SIGY	ETAN	BULK		
Type	I	F	F	F	F	F		
Default	none	none	none	none	0.0	none		

Card 2

Variable	EPF	PRF	REM	TREM				
Type	F	F	F	F				
Default	none	0.0	0.0	0.0				

VARIABLE	DESCRIPTION
MID	Material identification. A unique number has to be chosen.
RO	Mass density.
G	Shear modulus.
SIGY	Yield stress.
ETAN	Plastic hardening modulus.
BULK	Bulk modulus.
EPF	Plastic failure strain.
PRF	Failure pressure (≤ 0.0).

VARIABLE	DESCRIPTION
REM	Element erosion option: EQ.0.0: failed element eroded after failure, NE.0.0: element is kept, no removal except by Δt below.
TREM	Δt for element removal: EQ.0.0: Δt is not considered (default), GT.0.0: element eroded if element time step size falls below Δt.

When the effective plastic strain reaches the failure strain or when the pressure reaches the failure pressure, the element loses its ability to carry tension and the deviatoric stresses are set to zero, i.e., the material behaves like a fluid. If Δt for element removal is defined the element removal option is ignored.

The element erosion option based on Δt must be used cautiously with the contact options. Nodes to surface contact is recommended with all nodes of the eroded brick elements included in the node list. As the elements are eroded the mass remains and continues to interact with the master surface.

***MAT_SOIL_AND_FOAM_FAILURE**

This is Material Type 14. The input for this model is the same as for *MATERIAL_SOIL_AND_ FOAM (Type 5); however, when the pressure reaches the failure pressure, the element loses its ability to carry tension. It should be used only in situations when soils and foams are confined within a structure or when geometric boundaries are present.

*MAT_JOHNSON_COOK

This is Material Type 15. The Johnson/Cook strain and temperature sensitive plasticity is sometimes used for problems where the strain rates vary over a large range and adiabatic temperature increases due to plastic heating cause material softening.

Card Format

Card 1

Variable	MID	RO	G	E	PR	DTF		
Type	I	F	F	F	F	F		
Default	none	none	none	none	none	0.0		

Card 2

Variable	A	B	N	C	M	TM	TR	EPSO
Type	F	F	F	F	F	F	F	F
Default	none	0.0	0.0	0.0	none	none	none	none

Card 3

Variable	CP	PC	SPALL	IT	D1	D2	D3	D4
Type	F	F	F	F	F	F	F	F
Default	none	0.0	2.0	0.0	0.0	0.0	0.0	0.0

Card 4

Variable	D5							
Type	F							
Default	0.0							

VARIABLE	DESCRIPTION
MID	Material identification. A unique number has to be chosen.
RO	Mass density
G	Shear modulus
E	Young's Modulus (shell elements only)
PR	Poisson's ratio (shell elements only)
DTF	Minimum time step size for automatic element deletion (shell elements)
A	See equations below.
B	See equations below.
N	See equations below.
C	See equations below.
M	See equations below.
TM	Melt temperature
TR	Room temperature
EPSO	Effective plastic strain rate
CP	Specific heat
PC	Failure stress or pressure cutoff ($p_{min} < 0.0$)

VARIABLE	DESCRIPTION
SPALL	Spall type: EQ. 0.0: default set to "2.0", EQ. 1.0: $p \geq p_{min}$, EQ. 2.0: if $\sigma_{max} \geq -p_{min}$ element spalls and tension, $p < 0$, is never allowed, EQ. 3.0: $p < -p_{min}$ element spalls and tension, $p < 0$, is never allowed.
IT	Plastic strain iteration options: EQ. 0.0: no iterations (default), EQ. 1.0: accurate iterative solution for plastic strain. Much more expensive than default.
D1-D5	Failure parameters, see equations below.

Johnson and Cook express the flow stress as

$$\sigma_y = \left(A + B\,\bar{\varepsilon}^{\,p^n} \right)\left(1 + c\,\ln \dot{\varepsilon}^* \right)\left(1 - T^{*m} \right)$$

where

A, B, C, n, and m = input constants

$\bar{\varepsilon}^{\,p}$ effective plastic strain

$\dot{\varepsilon}^* = \dfrac{\dot{\bar{\varepsilon}}^{\,p}}{\dot{\varepsilon}_0}$ effective plastic strain rate for $\dot{\varepsilon}_0 = 1\,s^{-1}$

T^* = homologous temperature = $\dfrac{T - T_{room}}{T_{melt} - T_{room}}$

Constants for a variety of materials are provided in [Johnson and Cook 1983].

Due to nonlinearity in the dependence of flow stress on plastic strain, an accurate value of the flow stress requires iteration for the increment in plastic strain. However, by using a Taylor series expansion with linearization about the current time, we can solve for σ_y with sufficient accuracy to avoid iteration.

The strain at fracture is given by

$$\varepsilon^f = \left[D_1 + D_2 \exp D_3 \sigma^*\right]\left[1 + D_4 \ln \dot{\varepsilon}^*\right]\left[1 + D_5 T^*\right]$$

where σ^* is the ratio of pressure divided by effective stress

$$\sigma^* = \frac{p}{\sigma_{eff}}$$

Fracture occurs when the damage parameter

$$D = \sum \frac{\Delta \bar{\varepsilon}^p}{\varepsilon^f}$$

reaches the value of 1.

*MAT_PSEUDO_TENSOR

This is Material Type 16. This model has been used to analyze buried steel reinforced concrete structures subjected to implusive loadings.

Card Format

Card 1

Variable	MID	RO	G	PR				
Type	I	F	F	F				
Default	none	none	none	none				

Card 2

Variable	SIGF	A0	A1	A2	A0F	A1F	B1	PER
Type	F	F	F	F	F	F	F	F
Default	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0

Card 3

Variable	ER	PRR	SIGY	ETAN	LCP	LCR		
Type	F	F	F	F	F	F		
Default	0.0	0.0	none	0.0				

Card 4

Variable	EPS1	EPS2	EPS3	EPS4	EPS5	EPS6	EPS7	EPS8
Type	F	F	F	F	F	F	F	F
Default								

Card 5

Variable	EPS9	EPS10	EPS11	EPS12	EPS13	EPS14	EPS15	EPS16
Type	F	F	F	F	F	F	F	F
Default								

Card 6	1	2	3	4	5	6	7	8

Variable	ES1	ES2	ES3	ES4	ES5	ES6	ES7	ES8
Type	F	F	F	F	F	F	F	F
Default								

Card 7

Variable	ES9	ES10	ES11	ES12	ES13	ES14	ES15	ES16
Type	F	F	F	F	F	F	F	F
Default								

VARIABLE	DESCRIPTION
MID	Material identification. A unique number has to be chosen.
RO	Mass density.
G	Shear modulus.
PR	Poisson's ratio.
SIGF	Tensile cutoff (maximum principal stress for failure).
A0	Cohesion.
A1	Pressure hardening coefficient.
A2	Pressure hardening coefficient.
A0F	Cohesion for failed material.
A1F	Pressure hardening coefficient for failed material.
B1	Damage scaling factor.
PER	Percent reinforcement.
ER	Elastic modulus for reinforcement.
PRR	Poisson's ratio for reinforcement.
SIGY	Initial yield stress.
ETAN	Tangent modulus/plastic hardening modulus.
LCP	Load curve ID giving rate sensitivity for principal material, see *DEFINE_CURVE.
LCR	Load curve ID giving rate sensitivity for reinforcement, see *DEFINE_CURVE.
EPS	Effective plastic strain if A0 or A1 are nonzero. Otherwise, define pressure.
ES	Effective stress.

For the constant Poisson's ratio model, the shear modulus is computed from the bulk modulus. For the constant shear modulus model, Poisson's ratio is computed from the bulk modulus. The bulk modulus is determined by the equation of state.

If zero values are specified for a_0 and a_1, EPSi are taken to be pressure values instead of values of effective plastic strain.

If a negative value is specified for a_0, the value given for sigf is assumed to be the unconfined compressive strength of the principal material instead of the tensile cutoff value. In this case, values for the tensile cutoff and pressure hardening coefficients are calculated internally as follows:

$$
\begin{aligned}
\text{sigf} &= 1.7*((f'c)**2/\text{ucf})**(1/3) \\
a_0 &= (1/4) * (f'c) \\
a_1 &= (1/3) \\
a_2 &= (1/3)/ (f'c) \\
a_{0f} &= 0 \\
a_{1f} &= 0.385
\end{aligned}
$$

where ucf = $-a_0$ is a unit conversion factor for f′c (psi/(DYNA pressure unit)).

A zero equation of state number can also be specified in this case and data for a tri-linear E0S8 model (good for pressures below approximately 5 kbars) will be generated internally using the values given for Poisson's ratio and f′c. Otherwise, Equation of State 8, 9, or 11 must be specified and the corresponding data provided by the user.

Principal material and reinforcement properties are combined using a rule of mixtures as follows:

$$
\begin{aligned}
\text{bulk} &= (1\text{-fs})*\text{bkm}+\text{fs}*\text{bkr} \\
\text{shrm} &= (1\text{-fs})*\text{gm}+\text{fs}*\text{gr} \\
\text{sigy} &= (1\text{-fs})*\text{sym}+\text{fs}*\text{syr}
\end{aligned}
$$

where

$$\text{sym}=f\,(k1,\text{edot})*(a0+p/(a1+a2*p))*g(\text{dmg}) \text{ or } f(k1,\text{edot})*g(p)$$

and

$$\text{syr}=f\,(k2,\text{edot})*qs+qh*epx.$$

f (k,edot) denotes the yield stress strain-rate scaling factor obtained by linear interpolation from load curve k (if k=0, f=1.0), and g denotes either the damage or pressure scaling factor obtained by linear interpolation from the yield stress table. dmg is an isotropic measure of damage defined as

$$dmg = \int_0^{\bar{\varepsilon}^p} \frac{d\bar{\varepsilon}^p}{\left(1+\frac{p}{sigf}\right)**b1} \; ;$$

fs is the percent reinforcement, which is treated isotropically.

If the maximum principal stress in an element exceeds the tensile cutoff, the matrix material in that element is assumed to have fractured. After fracture, the matrix material in an element can support only compressive loads and its shear strength is limited by the yield surface for failed material

$$\text{symf} = a_{0f} + p'(a_{1f} + a_2 * p).$$

*MAT_ORIENTED_CRACK

This is Material Type 17. This material may be used to model brittle materials which fail due to large tensile stresses.

Card Format

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	E	PR	SIGY	ETAN	FS	PRF
Type	I	F	F	F	F	F	F	F
Default	none	none	none	none	none	0.0	none	0.0

VARIABLE	DESCRIPTION
MID	Material identification. A unique number has to be chosen.
RO	Mass density.
E	Young's modulus.
PR	Poisson's ratio.
SIGY	Yield stress.
ETAN	Plastic hardening modulus.
FS	Fracture stress.
PRF	Failure or cutoff pressure(≤ 0.0).

When the maximum principal stress exceeds the fracture stress, the element fails on a plane perpendicular to the direction of the maximum principal stress. In tension, the element will not carry any stresses on the fracture plane, but in compression, it will carry both normal and shear stresses. If the fracture stress is exceeded in another direction, the element fails isotropically: the element loses its ability to carry tension, the deviatoric stresses are set to zero, and the material behaves as a fluid.

*MAT_POWER_LAW_PLASTICITY

This is Material Type 18. This is an isotropic plasticity model with rate effects which uses a power law hardening rule.

Card Format

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	E	PR	K	N	SRC	SRP
Type	I	F	F	F	F	F	F	F
Default	none	none	none	none	none	none	0.0	0.0

VARIABLE	DESCRIPTION
MID	Material identification. A unique number has to be chosen.
RO	Mass density.
E	Young's modulus.
PR	Poisson's ratio.
K	Strength coefficient.
N	Hardening exponent.
SRC	Strain rate parameter, C, if zero, rate effects are ignored.
SRP	Strain rate parameter, P, if zero, rate effects are ignored.

Elastoplastic behavior with isotropic hardening is provided by this model. The yield stress, σ_y, is a function of plastic strain and obeys the equation:

$$\sigma_y = k\left(\varepsilon_e + \bar{\varepsilon}^p\right)^n$$

ε_e is the elastic strain to yield and $\bar{\varepsilon}^p$ is the effective plastic strain (logrithmic).

Strain rate is accounted for using the Cowper and Symonds model which scales the yield stress with the factor

$$1+\left(\frac{\dot{\varepsilon}}{C}\right)^{1/P}$$

where $\dot{\varepsilon}$ is the strain rate.

*MAT_STRAIN_RATE_DEPENDENT_PLASTICITY

This is Material Type 19. A strain rate dependent material can be defined. For an alternative, see Material Type 24. Required is a curve for the yield stress versus the effective strain rate. Optionally, Young's modulus and the tangent modulus can also be defined versus the effective strain rate. Also, optional failure of the material can be defined either by defining a von Mises stress at failure as a function of the effective strain rate (valid for solids/shells/thick shells) or by defining a minimum time step size (only for shells).

Card Format

Card 1

Variable	MID	RO	E	PR				
Type	I	F	F	F				
Default	none	none	none	none				

Card 2

Variable	LC1	ETAN	LC2	LC3	LC4	TDEL		
Type	F	F	F	F	F	F		
Default	none	0.0	0.0	0.0	0.0	0.0		

VARIABLE	DESCRIPTION
MID	Material identification. A unique number has to be chosen.
RO	Mass density.
E	Young's modulus.
PR	Poisson's ratio.
LC1	Load curve ID defining the yield stress σ_0 as a function of the effective strain rate.

VARIABLE	DESCRIPTION
ETAN	Plastic hardening modulus E_t
LC2	Load curve ID defining Young's modulus as a function of the effective strain rate (optional).
LC3	Load curve ID defining tangent modulus as a function of the effective strain rate (optional).
LC4	Load curve ID defining von Mises stress at failure as a function of the effective strain rate (optional).
TDEL	Minimum time step size for automatic element deletion. Use for shells only.

In this model, a load curve is used to describe the yield strength, σ_0, as a function of effective strain rate, $\dot{\bar{\varepsilon}}$, where

$$\dot{\bar{\varepsilon}} = \left(\frac{2}{3} \dot{\varepsilon}'_{ij} \dot{\varepsilon}'_{ij} \right)^{1/2}$$

and the prime denotes the deviatoric component. The yield stress is defined as

$$\sigma_y = \sigma_0 \left(\dot{\bar{\varepsilon}} \right) + E_h \, \bar{\varepsilon}^p$$

where $\bar{\varepsilon}^p$ is the effective plastic strain and E_h is given by

$$E_h = \frac{E \, E_t}{E - E_t} \ .$$

*MAT_RIGID

This is Material 20. Parts made from this material are considered to belong to a rigid body (for each part ID). Also, the coupling of a rigid body with MADYMO and CAL3D can be defined via this material. Alternatively, a VDA surface can be attached as surface to model the geometry, e.g, for the tooling in metalforming applications. Also, global and local constraints on the mass center can be optionally defined. Optionally, a local consideration for output and user-defined airbag sensors can be chosen.

Card Format

Card 1

Variable	MID	RO	E	PR	N	COUPLE	M	ALIAS
Type	I	F	F	F	F	F	F	C
Default	none	none	none	none	0	0	0	blank

Card 2

Variable	CMO	CON1	CON2					
Type	F	F	F					
Default	1.0	0	0					

Optional Card Format for output (Must be included but may be left blank).

Card 3

Variable	LCO or A1	A2	A3	V1	V2	V3		
Type	F	F	F	F	F	F		
Default	0	0	0	0	0	0		

VARIABLE	DESCRIPTION
MID	Material identification. A unique number has to be chosen.
RO	Mass density
E	Young's modulus. Reasonable values have to be chosen for contact analysis (choice of penalty), see remark below.
PR	Poisson's ratio. Reasonable values have to be chosen for contact analysis (choice of penalty), see remark below.
N	MADYMO3D **(not CAL3D)** coupling flag, n: EQ.0: use normal LS-DYNA3D rigid body updates, GT.0: the rigid body is coupled to MADYMO ellipsoid number n, LT.0: the rigid body is coupled to MADYMO plane number |n|.
COUPLE	Coupling option if applicable: EQ.-1: attach VDA surface in ALIAS (defined in the eighth field) and automatically generate a mesh for viewing the surface in LS-TAURUS. MADYMO3D/CAL3D coupling option: EQ.0: the undeformed geometry input to LS-DYNA3D corresponds to the local system for MADYMO/CAL3D. The finite element mesh is input, EQ.1: the undeformed geometry input to LS-DYNA3D corresponds to the global system for MADYMO/CAL3D, EQ.2: generate a mesh for the ellipsoids and planes internally in LS-DYNA3D.
M	MADYMO/CAL3D Coupling option flag: EQ.0: use normal LS-DYNA3D rigid body updates, EQ.m: this rigid body corresponds to MADYMO/CAL3D rigid body number m. Rigid body updates are performed by MADYMO/CAL3D.
ALIAS	VDA surface alias name, see Appendix I.

VARIABLE	DESCRIPTION
CMO	Center of mass constraint option, CMO: EQ.+1.0: constraints applied in global directions, EQ. -1.0: constraints applied in local directions (SPC constraint).
CON1	First constraint parameter: If CMO=+1.0, then specify global translational constraint: EQ.0: no constraints, EQ.1: constrained x displacement, EQ.2: constrained y displacement, EQ.3: constrained z displacement, EQ.4: constrained x and y displacements, EQ.5: constrained y and z displacements, EQ.6: constrained z and x displacements, EQ.7: constrained x, y, and z displacements. If CM0=-1.0, then specify local coordinate system ID. See *DEFINE_ COORDINATE_OPTION:
CON2	Second constraint parameter: If CMO=+1.0, then specify global rotational constraint: EQ.0: no constraints, EQ.1: constrained x rotation, EQ.2: constrained y rotation, EQ.3: constrained z rotation, EQ.4: constrained x and y rotations, EQ.5: constrained y and z rotations, EQ.6: constrained z and x rotations, EQ.7: constrained x, y, and z rotations. If CM0=-1.0, then specify local (SPC) constraint: EQ.000000 no constraint, EQ.100000 constrained x translation, EQ.010000 constrained y translation, EQ.001000 constrained z translation, EQ.000100 constrained x rotation, EQ.000010 constrained y rotation, EQ.000001 constrained z rotation. Any combination of local constraints can be achieved by adding the number 1 into the corresponding column.
LCO	Local coordinate system number for output. See *DEFINE_ COORDINATE_OPTION. *****Alternative method for specifying local system below.*****
A1-V3	Define two vectors **a** and **v**, fixed in the rigid body which are used for output and the user defined airbag sensor subroutines. The output parameters are in the directions **a**, **b**, and **c** where the latter are given by the cross products **c=a**×**v** and **b=c**×**a.** This input is optional.

Remark:

1. The material constants are used for determining sliding interface parameters if the rigid body interacts along sliding interfaces. Realistic values for these constants should be defined.

*MAT_ORTHOTROPIC_THERMAL

This is Material Type 21. A linearly elastic material with orthotropic temperature dependent coefficients can be defined.

Card Format

Card 1

Variable	MID	RO	EA	EB	EC	PRBA	PRCA	PRCB
Type	I	F	F	F	F	F	F	F

Card 2

Variable	GAB	GBC	GCA	AA	AB	AC	AOPT	
Type	F	F	F	F	F	F	F	

Card 3

Variable	XP	YP	ZP	A1	A2	A3		
Type	F	F	F	F	F	F		

Card 4

Variable	V1	V2	V3	D1	D2	D3	BETA	
Type	F	F	F	F	F	F	F	

VARIABLE	DESCRIPTION
MID	Material identification. A unique number has to be chosen.
RO	Mass density.
EA	E_a, Young's modulus in a-direction.
EB	E_b, Young's modulus in b-direction.
EC	E_c, Young's modulus in c-direction.
PRBA	ν_{ba}, Poisson's ratio, ba.
PRCA	ν_{ca}, Poisson's ratio, ca.
PRCB	ν_{cb}, Poisson's ratio, cb
GAB	G_{ab}, Shear modulus, ab.
GBC	G_{bc}, Shear modulus, bc.
GCA	G_{ca}, Shear modulus, ca.
AA	α_a, coefficients of thermal expansion in the a-direction.
AB	α_b, coefficients of thermal expansion in the b-direction.
AC	α_c, coefficients of thermal expansion in the c-direction.
AOPT	Material axes option, see Figure 19.1: EQ. 0.0: locally orthotropic with material axes determined by element nodes as shown in Figure 19.1. Nodes 1, 2, and 4 of an element are identical to the Nodes used for the definition of a coordinate system as by *DEFINE_COORDINATE_NODES. EQ. 1.0: locally orthotropic with material axes determined by a point in space and the global location of the element center, this is the a-direction. EQ. 2.0: globally orthotropic with material axes determined by vectors defined below, as with *DEFINE_COORDINATE_VECTOR. EQ. 3.0: applicable to shell elements only. This option determines locally orthotropic material axes by offsetting the material axes by an angle to be specified from a line in the plane of the shell determined by taking the cross product of the vector **v** defined below with the shell normal vector. EQ. 4.0: locally orthotropic in cylindrical coordinate system with the material axes determined by a vector **v** and an orginating point, P. They define the axis of symmetry.
XP,YP,ZP	Coordinates of point **p** for AOPT = 1.
A1,A2,A3	Components of vector **a** for AOPT = 2.

VARIABLE	DESCRIPTION
V1,V2,V3	Components of vector **v** for AOPT = 3.
D1,D2,D3	Components of vector **d** for AOPT = 2.
BETA	Material angle for AOPT = 3, may be overridden on the element card. This angle is measured with respect to y=v x n.

***MAT_COMPOSITE_DAMAGE**

This is Material Type 22. An orthotropic material with optional brittle failure for composites can be defined following the suggestion of [Chang and Chang 1982a,1982b]. Three failure criteria are possible, see Theoretical Manual. After bulk compression, a force can be still transmitted, however, another bulk modulus has to be given.

Card Format

Card 1

Variable	MID	RO	EA	EB	EC	PRBA	PRCA	PRCB
Type	I	F	F	F	F	F	F	F
Default	none	none	none	none	none	none	none	none

Card 2

Variable	GAB	GBC	GCA	KFAIL	AOPT	MACF		
Type	F	F	F	F	F	F		
Default	none	none	none	0.0	0.0	1.0		

Card 3

Variable	XP	YP	ZP	A1	A2	A3		
Type	F	F	F	F	F	F		
Default	0.0	0.0	0.0	0.0	0.0	0.0		

Card 4

Variable	V1	V2	V3	D1	D2	D3		
Type	F	F	F	F	F	F		
Default	0.0	0.0	0.0	0.0	0.0	0.0		

Card 5

Variable	SC	XT	YT	YC	ALPH			
Type	F	F	F	F	F			
Default	none	none	none	none	none			

VARIABLE	DESCRIPTION
MID	Material identification. A unique number has to be chosen.
RO	Mass density
EA	E_a, Young's modulus in a-direction.
EB	E_b, Young's modulus in b-direction.
EC	E_c, Young's modulus in c-direction.
PRBA	ν_{ba}, Poisson ratio, ba.
PRCA	ν_{ca}, Poisson ratio, ca.
PRCB	ν_{cb}, Poisson ratio, cb.
GAB	G_{ab}, Shear modulus, ab.
GBC	G_{bc}, Shear modulus, bc.
GCA	G_{ca}, Shear modulus, ca.
KFAIL	Bulk modulus of failed material. Necessary for compressive failure.

VARIABLE	DESCRIPTION
AOPT	Material axes option: EQ. 0.0: locally orthotropic with material axes determined by element nodes as shown in Figure 19.1. Nodes 1, 2, and 4 of an element are identical to the Nodes used for the definition of a coordinate system as by *DEFINE_COORDINATE_NODES. EQ. 1.0: locally orthotropic with material axes determined by a point in space and the global location of the element center, this is the a-direction. EQ. 2.0: globally orthotropic with material axes determined by vectors defined below, as with *DEFINE_COORDINATE_VECTOR. EQ. 3.0: applicable to shell elements only. This option determines locally orthotropic material axes by offsetting the material axes by an angle to be specified from a line in the plane of the shell determined by taking the cross product of the vector **v** defined below with the shell normal vector. EQ. 4.0: locally orthotropic in cylindrical coordinate system with the material axes determined by a vector **v** and an orginating point, P. They define the axis of symmetry.
MACF	Material axes change flag for brick elements: EQ.1.0: default, EQ.2.0: switch material axes a and b, EQ.3.0: switch material axes a and c.
XP,YP,ZP	Coordinates of point **p** for AOPT = 1.
A1,A2,A3	Components of vector **a** for AOPT = 2.
V1,V2,V3	Components of vector **v** for AOPT = 3.
D1,D2,D3	Components of vector **d** for AOPT = 2.
SC	Shear strength, ab plane, see Theoretical Manual.
XT	Longitudinal tensile strength, a-axis, see Theoretical Manual.
YT	Transverse tensile strength, b-axis, see Theoretical Manual.
YC	Transverse compressive strength, b-axis, see Theoretical Manual.
ALPH	Shear stress parameter for the nonlinear term, see Theoretical Manual. Suggested range 0 – 0.5.

*MAT_TEMPERATURE_DEPENDENT_ORTHOTROPIC

This is Material Type 23. An orthotropic elastic material with arbitrary temperature dependency can be defined.

Card Format

Card 1

1	2	3	4	5	6	7	8
Variable	MID	RO	AOPT					
Type	I	F	F					

Card 2

Variable	XP	YP	ZP	A1	A2	A3		
Type	F	F	F	F	F	F		

Card 3

Variable	V1	V2	V3	D1	D2	D3		
Type	F	F	F	F	F	F		

Define one set of constants on two cards for each temperature point.
Up to 48 points (96 cards) can defined. The next "*" card terminates the input.

Cards 1 for Temperature Ti

Variable	EAi	EBi	ECi	PRABi	PRCAi	PRCBi		
Type	F	F	F	F	F	F		

Cards 2 for Temperature Ti

Variable	AAi	ABi	ACi	GABi	GCAi	GCBi	Ti	
Type	F	F	F	F	F	F	F	

VARIABLE	DESCRIPTION
MID	Material identification. A unique number has to be chosen.
RO	Mass density.
AOPT	Material axes option: EQ. 0.0: locally orthotropic with material axes determined by element nodes as shown in Figure 19.1. Nodes 1, 2, and 4 of an element are identical to the Nodes used for the definition of a coordinate system as by *DEFINE_COORDINATE_NODES. EQ. 1.0: locally orthotropic with material axes determined by a point in space and the global location of the element center, this is the a-direction. EQ. 2.0: globally orthotropic with material axes determined by vectors defined below, as with *DEFINE_COORDINATE_VECTOR. EQ. 3.0: applicable to shell elements only. This option determines locally orthotropic material axes by offsetting the material axes by an angle to be specified from a line in the plane of the shell determined by taking the cross product of the vector **v** defined below with the shell normal vector. EQ. 4.0: locally orthotropic in cylindrical coordinate system with the material axes determined by a vector **v** and an orginating point, P. They define the axis of symmetry.
XP,YP,ZP	Coordinates of point **p** for AOPT = 1.
A1,A2,A3	Components of vector **a** for AOPT = 2.
V1,V2,V3	Components of vector **v** for AOPT = 3.
D1,D2,D3	Components of vector **d** for AOPT = 2.
EAi	E_a, Young's modulus in a-direction at temperature Ti.
EBi	E_b, Young's modulus in b-direction at temperature Ti.
ECi	E_c, Young's modulus in c-direction at temperature Ti.

VARIABLE	DESCRIPTION
PRBAi	ν_{ba}, Poisson's ratio ba at temperature Ti.
PRCAi	ν_{ca}, Poisson's ratio ca at temperature Ti.
PRCBi	ν_{cb}, Poisson's ratio cb at temperature Ti.
AAi	α_a, coefficient of thermal expansion in a-direction at temperature Ti.
ABi	α_b, coefficient of thermal expansion in b-direction at temperature Ti.
ACi	α_c, coefficient of thermal expansion in c-direction at temperature Ti.
GABi	G_{ab}, Shear modulus ab at temperature Ti.
GBCi	G_{bc}, Shear modulus bc at temperature Ti.
GCAi	G_{ca}, Shear modulus ca at temperature Ti.
Ti	ith temperature

*MAT_PIECEWISE_LINEAR_PLASTICITY

This is Material Type 24. An elasto-plastic material with an arbitrary stress versus strain curve and arbitrary strain rate dependency can be defined. See also Remark below. Also, failure based on a plastic strain or a minimum time step size can be defined.

Card Format

Card 1

Variable	MID	RO	E	PR	SIGY	ETAN	EPPF	TDEL
Type	I	F	F	F	F	F	F	F
Default	none	none	none	none	none	0.0	10.E+20	10.E+20

Card 2

Variable	C	P	LCSS	LCSR				
Type	F	F	F	F				
Default	0	0	0	0				

Card 3

Variable	EPS1	EPS2	EPS3	EPS4	EPS5	EPS6	EPS7	EPS8
Type	F	F	F	F	F	F	F	F
Default	0	0	0	0	0	0	0	0

Card 4

Variable	ES1	ES2	ES3	ES4	ES5	ES6	ES7	ES8
Type	F	F	F	F	F	F	F	F
Default	0	0	0	0	0	0	0	0

VARIABLE	DESCRIPTION
MID	Material identification. A unique number has to be chosen.
RO	Mass density.
E	Young's modulus.
PR	Poisson's ratio.
SIGY	Yield stress.
ETAN	Tangent modulus, ignored if (LCSS.GT.0) is defined.
EPPF	Plastic strain at failure (logrithmic).
TDEL	Minimum time step size for automatic element deletion.
C	Strain rate parameter, C, see formula below.
P	Strain rate parameter, P, see formula below.
LCSS	Load curve ID or Table ID. Load curve ID defining effective stress versus effective plastic strain. If defined EPS1-EPS8 and ES1-ES8 are ignored. The table ID defines for each strain rate value a load curve ID giving the stress versus effectiveplastic strain for that rate, See Figure 19.5. The stress versus effective plastic strain curve for the lowest value of strain rate is used if the strain rate falls below the minimum value. Likewise, the stress versus effective plastic strain curve for the highest value of strain rate is used if the strain rate exceeds the maximum value. The strain rate parameters: C and P; the curve ID, LCSR; EPS1-EPS8 and ES1-ES8 are ignored if a Table ID is defined.
LCSR	Load curve ID defining strain rate scaling effect on yield stress.

VARIABLE	DESCRIPTION
EPS1-EPS8	Effective plastic strain values (optional if SIGY is defined). At least 2 points should be defined.
ES1-ES8	Corresponding yield stress values to EPS1 - EPS8.

The stress strain behavior may be treated by a bilinear stress strain curve by defining the tangent modulus, ETAN. Alternately, a curve similar to that shown in Figure 19.4 is expected to be defined by (EPS1,ES1) - (EPS8,ES8); however, an effective stress versus effective plastic strain curve (LCSS) may be input instead if eight points are insufficient. The cost is roughly the same for either approach. The most general approach is to use the table definition (LCSS) discussed below.

Three options to account for strain rate effects are possible.

I. Strain rate may be accounted for using the Cowper and Symonds model which scales the yield stress with the factor

$$1+\left(\frac{\dot{\varepsilon}}{C}\right)^{1/p}$$

where $\dot{\varepsilon}$ is the strain rate. $\dot{\varepsilon}=\sqrt{\dot{\varepsilon}_{ij}\dot{\varepsilon}_{ij}}$

II. For complete generality a load curve (LCSR) to scale the yield stress may be input instead. In this curve the scale factor versus strain rate is defined.

III. If different stress versus strain curves can be provided for various strain rates, the option using the reference to a table (LCSS) can be used. Then the table input in *DEFINE_TABLE has to be used, see Figure 19.5.

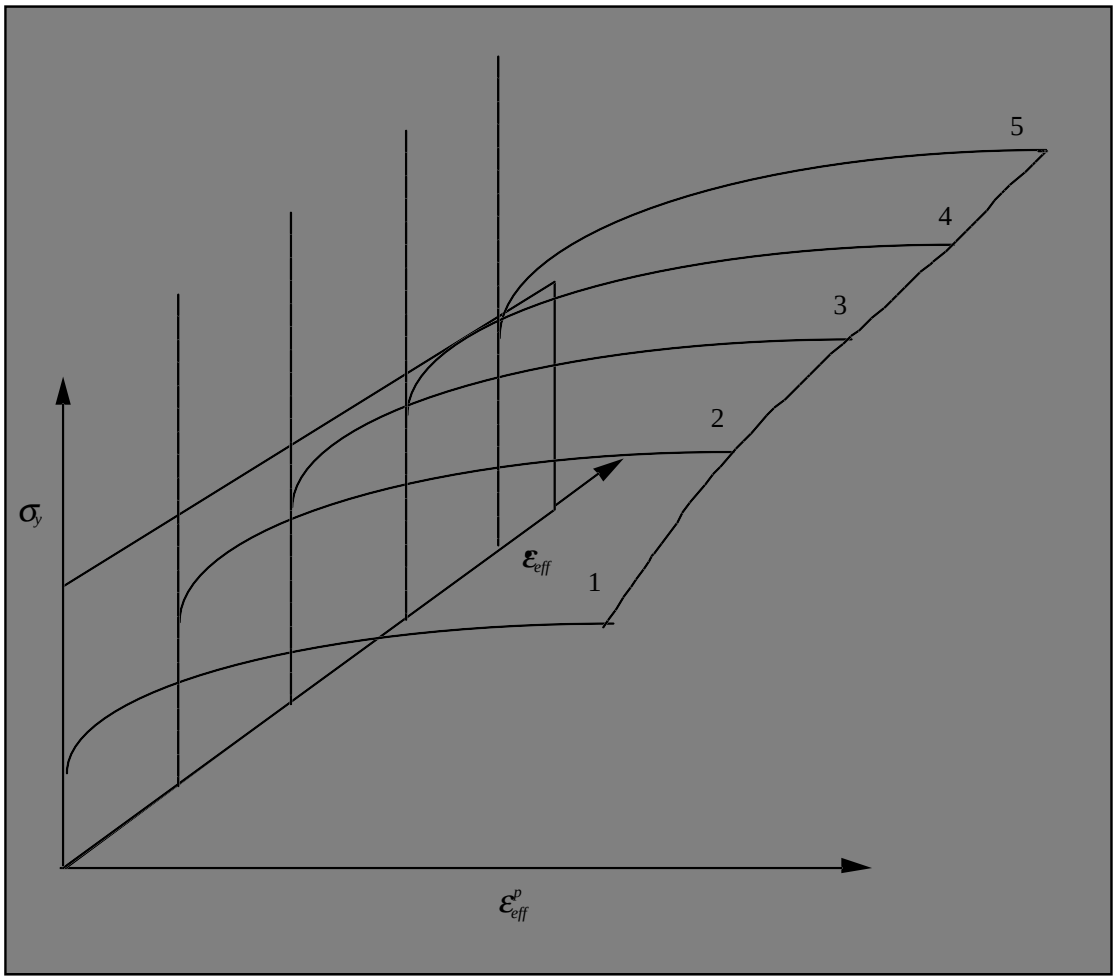


Figure 19.5. Rate effects may be accounted for by defining a table of curves. If a table ID is specified a curve ID is given for each strain rate, see *DEFINE_TABLE. Intermediate values are found by interpolating between curves. Effective plastic strain versus yield stress is expected.

*MAT_GEOLOGIC_CAP_MODEL

This is Material Type 25. This an inviscid two invariant geologic cap model. This material model can be used for geomechanical problems or for materials as concrete, see references cited below.

Card Format

Card 1

Variable	MID	RO	BULK	G	ALPHA	THETA	GAMMA	BETA
Type	I	F	F	F	F	F	F	F

Card 2

Variable	R	D	W	X0	C	N		
Type	F	F	F	F	F	F		

Card 3

Variable	PLOT	FTYPE	VEC	TOFF				
Type	F	F	F	F				

VARIABLE	DESCRIPTION
MID	Material identification. A unique number has to be chosen.
RO	Mass density.
BULK	Initial bulk modulus, K.
G	Initial Shear modulus.
ALPHA	Failure envelope parameter, α.

VARIABLE	DESCRIPTION
THETA	Failure envelope linear coefficient, θ.
GAMMA	Failure envelope exponential coefficient, γ.
BETA	Failure envelope exponent, β.
R	Cap, surface axis ratio.
D	Hardening law exponent.
W	Hardening law coefficient.
X0	Hardening law exponent, X_0.
C	Kinematic hardening coefficient, $\bar{c}$.
N	Kinematic hardening parameter.
PLOT	Save the following variable for plotting in TAURUS, to be labeled there as "effective plastic strain:" EQ.1: hardening parameter, κ, EQ.2: cap -J_1 axis intercept, X (κ), EQ.3: volumetric plastic strain ε_V^p , EQ.4: first stress invarient, J_1, EQ.5: second stress invarient, $\sqrt{J_2}$, EQ.6: not used, EQ.7: not used, EQ.8: response mode number, EQ.9: number of iterations.
FTYPE	Formulation flag: EQ.1: soil or concrete (Cap surface may contract), EQ.2: rock (Cap doesn't contract).
VEC	Vectorization flag: EQ.0: vectorized (fixed number of iterations), EQ.1: fully iterative, If the vectorized solution is chosen, the stresses might be slightly off the yield surface; however, on vector computers a much more efficient solution is achieved.
TOFF	Tension Cut Off, TOFF < 0 (positive in compression).

The implementation of an extended two invariant cap model, suggested by Stojko [1990], is based on the formulations of Simo, et. al. [1988, 1990] and Sandler and Rubin [1979]. In this model, the two invariant cap theory is extended to include nonlinear kinematic hardening as suggested by Isenberg, Vaughn, and Sandler [1978]. A brief discussion of the extended cap model and its parameters is given below.

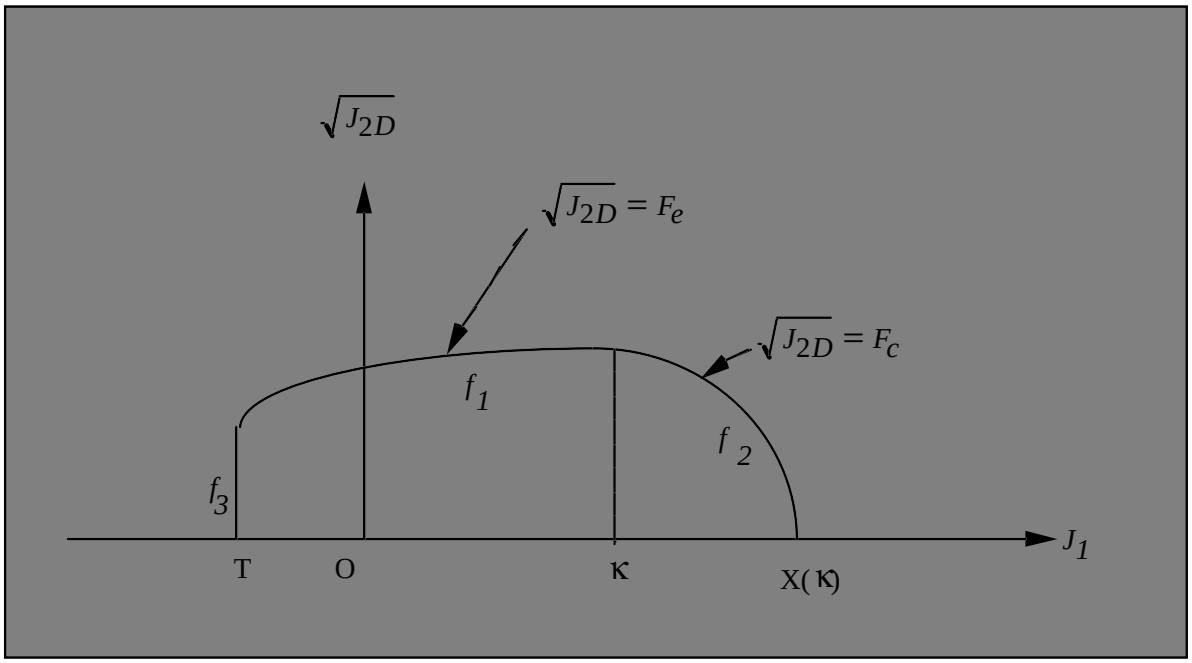


Figure 19.6. The yield surface of the two-invariant cap model in pressure $\sqrt{J_{2D}} - J_1$ space. Surface f_1 is the failure envelope, f_2 is the cap surface, and f_3 is the tension cutoff.

The cap model is formulated in terms of the invariants of the stress tensor. The square root of the second invariant of the deviatoric stress tensor, $\sqrt{J_{2D}}$ is found from the deviatoric stresses **s** as

$$\sqrt{J_{2D}} \equiv \sqrt{\frac{1}{2} s_{ij} s_{ij}}$$

and is the objective scalar measure of the distortional or shearing stress. The first invariant of the stress, J_1, is the trace of the stress tensor.

The cap model consists of three surfaces in $\sqrt{J_{2D}} - J_1$ space, as shown in Figure 19.6. First, there is a failure envelope surface, denoted f_1 in the figure. The functional form of f_1 is

$$f_1 = \sqrt{J_{2D}} - \min\left(F_e(J_1), T_{mises}\right) ,$$

where F_e is given by

$$F_e(J_1) \equiv \alpha - \gamma \exp(-\beta J_1) + \theta J_1$$

and $T_{mises} \equiv \left| X(\kappa_n) - L(\kappa_n) \right|$. This failure envelop surface is fixed in $\sqrt{J_{2D}} - J_1$ space, and therefore does not harden unless kinematic hardening is present. Next, there is a cap surface, denoted f_2 in the figure, with f_2 given by

$$f_2 = \sqrt{J_{2D}} - F_c(J_1, \kappa)$$

where F_c is defined by

$$F_c(J_1, \kappa) \equiv \frac{1}{R}\sqrt{\left[X(\kappa) - L(\kappa)\right]^2 - \left[J_1 - L(\kappa)\right]^2},$$

$X(\kappa)$ is the intersection of the cap surface with the J_1 axis

$$X(\kappa) = \kappa + RF_e(\kappa),$$

and $L(\kappa)$ is defined by

$$L(\kappa) \equiv \begin{cases} \kappa \text{ if } \kappa > 0 \\ 0 \text{ if } \kappa \le 0 \end{cases}.$$

The hardening parameter κ is related to the plastic volume change ε_v^p through the hardening law

$$\varepsilon_v^p = W\left\{1 - \exp\left[-D\left(X(\kappa) - X_0\right)\right]\right\}$$

Geometrically, κ is seen in the figure as the J_1 coordinate of the intersection of the cap surface and the failure surface. Finally, there is the tension cutoff surface, denoted f_3 in the figure. The function f_3 is given by

$$f_3 \equiv T - J_1,$$

where T is the input material parameter which specifies the maximum hydrostatic tension sustainable by the material. The elastic domain in $\sqrt{J_{2D}} - J_1$ space is then bounded by the failure envelope surface above, the tension cutoff surface on the left, and the cap surface on the right.

An additive decomposition of the strain into elastic and plastic parts is assumed:

$$\boldsymbol{\varepsilon} = \boldsymbol{\varepsilon}^e + \boldsymbol{\varepsilon}^p,$$

where $\boldsymbol{\varepsilon}^e$ is the elastic strain and $\boldsymbol{\varepsilon}^p$ is the plastic strain. Stress is found from the elastic strain using Hooke's law,

$$\boldsymbol{\sigma} = \mathbf{C}(\boldsymbol{\varepsilon} - \boldsymbol{\varepsilon}^p) ,$$

where $\boldsymbol{\sigma}$ is the stress and $\mathbf{C}$ is the elastic constitutive tensor.

The yield condition may be written

$$f_1(\boldsymbol{\sigma}) \le 0$$
$$f_2(\boldsymbol{\sigma}, \kappa) \le 0$$
$$f_3(\boldsymbol{\sigma}) \le 0$$

and the plastic consistency condition requires that

$$\dot{\lambda}_k \; f_k = 0 \qquad k = 1,2,3$$
$$\dot{\lambda}_k \ge 0$$

where λ_k is the plastic consistency parameter for surface k. If $f_k < 0$ then, $\dot{\lambda}_k = 0$ and the response is elastic. If $f_k > 0$ then surface k is active and $\dot{\lambda}_k$ is found from the requirement that $\dot{f}_k = 0$.

Associated plastic flow is assumed, so using Koiter's flow rule the plastic strain rate is given as the sum of contribution from all of the active surfaces,

$$\dot{\boldsymbol{\varepsilon}}^p = \sum_{k=1}^{3} \dot{\lambda}_k \frac{\partial f_k}{\partial \boldsymbol{\sigma}} \quad .$$

One of the major advantages of the cap model over other classical pressure-dependent plasticity models is the ability to control the amount of dilatency produced under shear loading. Dilatency is produced under shear loading as a result of the yield surface having a positive slope in $\sqrt{J_{2D}} - J_1$ space, so the assumption of plastic flow in the direction normal to the yield surface produces a plastic strain rate vector that has a component in the volumetric (hydrostatic) direction (see Figure 19.6). In models such as the Drucker-Prager and Mohr-Coulomb, this dilatency continues as long as shear loads are applied, and in many cases produces far more dilatency than is experimentally observed in material tests. In the cap model, when the failure surface is active, dilatency is produced just as with the Drucker-Prager and Mohr-Columb models. However, the hardening law permits the cap surface to contract until the cap intersects the failure envelope at the stress point, and the cap remains at that point. The local normal to the yield surface is now vertical,

and therefore the normality rule assures that no further plastic volumetric strain (dilatency) is created. Adjustment of the parameters that control the rate of cap contractions permits experimentally observed amounts of dilatency to be incorporated into the cap model, thus producing a constitutive law which better represents the physics to be modeled.

Another advantage of the cap model over other models such as the Drucker-Prager and Mohr-Coulomb is the ability to model plastic compaction. In these models all purely volumetric response is elastic. In the cap model, volumetric response is elastic until the stress point hits the cap surface. Therefore, plastic volumetric strain (compaction) is generated at a rate controlled by the hardening law. Thus, in addition to controlling the amount of dilatency, the introduction of the cap surface adds another experimentally observed response characteristic of geological material into the model.

The inclusion of kinematic hardening results in hysteretic energy dissipation under cyclic loading conditions. Following the approach of Isenberg, et. al. [1978] a nonlinear kinematic hardening law is used for the failure envelope surface when nonzero values of and N are specified. In this case, the failure envelope surface is replaced by a family of yield surfaces bounded by an initial yield surface and a limiting failure envelope surface. Thus, the shape of the yield surfaces described above remains unchanged, but they may translate in a plane orthogonal to the J axis,

Translation of the yield surfaces is permitted through the introduction of a "back stress" tensor, $\boldsymbol{\alpha}$. The formulation including kinematic hardening is obtained by replacing the stress $\boldsymbol{\sigma}$ with the translated stress tensor $\eta \equiv \sigma - \alpha$ in all of the above equation. The history tensor $\boldsymbol{\alpha}$ is assumed deviatoric, and therefore has only 5 unique components. The evolution of the back stress tensor is governed by the nonlinear hardening law

$$\alpha = \bar{c}\bar{F}(\sigma, \alpha)\dot{e}^{p}$$

where $\bar{c}$ is a constant, $\bar{F}$ is a scalar function of $\boldsymbol{\sigma}$ and $\boldsymbol{\alpha}$ and $\dot{e}^{p}$ is the rate of deviator plastic strain. The constant may be estimated from the slope of the shear stress - plastic shear strain curve at low levels of shear stress.

The function $\bar{F}$ is defined as

$$\bar{F} \equiv \max\left(0{,}1 - \frac{(\sigma - \alpha) \bullet \alpha}{2NF_e(J_1)}\right)$$

where N is a constant defining the size of the yield surface. The value of N may be interpreted as the radial distant between the outside of the initial yield surface and the inside of the limit surface. In order for the limit surface of the kinematic hardening cap model to correspond with the failure

envelope surface of the standard cap model, the scalar parameter a must be replaced α - N in the definition F_e.

The cap model contains a number of parameters which must be chosen to represent a particular material, and are generally based on experimental data. The parameters α, β, θ, and γ are usually evaluated by fitting a curve through failure data taken from a set of triaxial compression tests. The parameters W, D, and X_0 define the cap hardening law. The value W represent the void fraction of the uncompressed sample and D governs the slope of the initial loading curve in hydrostatic compression. The value of R is the ration of major to minor axes of the quarter ellipse defining the cap surface. Additional details and guidelines for fitting the cap model to experimental data are found in (Chen and Baladi, 1985).

*MAT_HONEYCOMB

This is Material Type 26. The major use of this material model is for honeycomb and foam materials with real anisotropic behavior. A nonlinear elastoplastic material behavior can be defined separately for all normal and shear stresses. These are considered to be fully uncoupled. See notes below.

Card Format

Card 1

Variable	MID	RO	E	PR	SIGY	VF	MU	BULK
Type	I	F	F	F	F	F	F	F
Default	none	none	none	none	none	none	.05	0.0

Card 2

Variable	LCA	LCB	LCC	LCS	LCAB	LCBC	LCCA	LCSR
Type	F	F	F	F	F	F	F	F
Default	none	LCA	LCA	LCA	LCS	LCS	LCS	optional

Card 3

Variable	EAAU	EBBU	ECCU	GABU	GBCU	GCAU	AOPT	
Type	F	F	F	F	F	F		

Card 4

Variable	XP	YP	ZP	A1	A2	A3		
Type	F	F	F	F	F	F		

Card 5

Variable	D1	D2	D3					
Type	F	F	F					

VARIABLE	DESCRIPTION
MID	Material identification. A unique number has to be chosen.
RO	Mass density.
E	Young's modulus for compacted honeycomb material.
PR	Poisson's ratio for compacted honeycomb material.
SIGY	Yield stress for fully compacted honeycomb.
VF	Relative volume at which the honeycomb is fully compacted.
MU	μ, material viscosity coefficient. (default=.05) Recommended.
BULK	Bulk viscosity flag: EQ.0.0: bulk viscosity is not used. This is recommended. EQ.1.0: bulk viscosity is active and μ=0 This will give results identical to previous versions of LS-DYNA3D.
LCA	Load curve ID, see *DEFINE_CURVE, for sigma-aa versus either relative volume or volumetric strain. See notes below.
LCB	Load curve ID, see *DEFINE_CURVE, for sigma-bb versus either relative volume or volumetric strain. Default LCB=LCA. See notes below.
LCC	Load curve ID, see *DEFINE_CURVE, for sigma-cc versus either relative volume or volumetric strain. Default LCC=LCA. See notes below.
LCS	Load curve ID, see *DEFINE_CURVE, for shear stress versus either relative volume or volumetric strain. Default LCS=LCA. Each component of shear stress may have its own load curve. See notes below.
LCAB	Load curve ID, see *DEFINE_CURVE, for sigma-ab versus either relative volume or volumetric strain. Default LCAB=LCS. See notes below.

VARIABLE	DESCRIPTION
LCBC	Load curve ID, see *DEFINE_CURVE, for sigma-bc versus either relative volume or volumetric strain. Default LCBC=LCS. See notes below.
LCCA	Load curve ID, see *DEFINE_CURVE, or sigma-ca versus either relative volume or volumetric strain. Default LCCA=LCS. See notes below.
LCRS	Load curve ID, see *DEFINE_CURVE, for strain-rate effects defining the scale factor versus strain rate $\varepsilon = \sqrt{\ }$. This is optional. The curves defined above are scaled using this curve.
EAAU	Elastic modulus E_{aau} in uncompressed configuration.
EBBU	Elastic modulus E_{bbu} in uncompressed configuration.
ECCU	Elastic modulus E_{ccu} in uncompressed configuration.
GABU	Shear modulus G_{abu} in uncompressed configuration.
GBCU	Shear modulus G_{bcu} in uncompressed configuration.
GCAU	Shear modulus G_{cau} in uncompressed configuration.
AOPT	Material axes option: EQ. 0.0: locally orthotropic with material axes determined by element nodes as shown in Figure 19.1. Nodes 1, 2, and 4 of an element are identical to the Nodes used for the definition of a coordinate system as by *DEFINE_COORDINATE_NODES. EQ. 1.0: locally orthotropic with material axes determined by a point in space and the global location of the element center, this is the a-direction. EQ. 2.0: globally orthotropic with material axes determined by vectors defined below, as with *DEFINE_COORDINATE_VECTOR. EQ. 3.0: This option determines locally orthotropic material axes by offsetting the material axes by an angle to be specified from a line in the plane of the 1-2-3-4 segment determined by taking the cross product of the vector **v** defined below with the segment normal vector.
XP YP ZP	Coordinates of point p for AOPT = 1.
A1 A2 A3	Components of vector a for AOPT = 2.
D1 D2 D3	Components of vector d for AOPT = 2.

For efficiency it is strongly recommended that the load curve ID's: LCA, LCB, LCC, LCS, LCAB, LCBC, and LCCA, contain exactly the same number of points with corresponding strain values on the abcissa. If this recommendation is followed the cost of the table lookup is insignificant. Conversely, the cost increases significantly if the abcissa strain values are not consistent between load curves.

The behavior before compaction is orthotropic where the components of the stress tensor are uncoupled, i.e., an a component of strain will generate resistance in the local *a*-direction with no coupling to the local *b* and *c* directions. The elastic moduli vary, from their initial values to the fully compacted values at V_f, linearly with the relative volume V:

$$E_{aa} = E_{aau} + \beta(E - E_{aau})$$

$$E_{bb} = E_{bbu} + \beta(E - E_{bbu})$$

$$E_{cc} = E_{ccu} + \beta(E - E_{ccu})$$

$$G_{ab} = G_{abu} + \beta(G - G_{abu})$$

$$G_{bc} = G_{bcu} + \beta(G - G_{bcu})$$

$$G_{ca} = G_{cau} + \beta(G - G_{cau})$$

where

$$\beta = \max\left[\ \min\left(\frac{1-V}{1-V_f},1\right),0\right]$$

and G is the elastic shear modulus for the fully compacted honeycomb material

$$G = \frac{E}{2(1+\nu)}\ .$$

The relative volume, V, is defined as the ratio of the current volume to the initial volume. Typically, V=1 at the beginning of a calculation. The viscosity coefficient μ (MU) should be set to a small number (usually .02-.10 is okay). Alternatively, the two bulk viscosity coefficients on the control cards should be set to very small numbers to prevent the development of spurious pressures that may lead to undesirable and confusing results. The latter is not recommended since spurious numerical noise may develop.

The load curves define the magnitude of the average stress as the material changes density (relative volume), see Figure 19.7. Each curve related to this model must have the same number of points and the same abscissa values. There are two ways to define these curves, **a)** as a function of relative volume (V) or **b)** as a function of volumetric strain defined as:

$$\varepsilon_v = 1 - V$$

In the former, the first value in the curve should correspond to a value of relative volume slightly less than the fully compacted value. In the latter, the first value in the curve should be less than or equal to zero, corresponding to tension, and increase to full compaction. **Care should be taken when defining the curves so that extrapolated values do not lead to negative yield stresses.**

At the beginning of the stress update each element's stresses and strain rates are transformed into the local element coordinate system. For the uncompacted material, the trial stress components are updated using the elastic interpolated moduli according to:

$$\sigma_{aa}^{n+1^{trial}} = \sigma_{aa}^{n} + E_{aa}\Delta\varepsilon_{aa}$$

$$\sigma_{bb}^{n+1^{trial}} = \sigma_{bb}^{n} + E_{bb}\Delta\varepsilon_{bb}$$

$$\sigma_{cc}^{n+1^{trial}} = \sigma_{cc}^{n} + E_{cc}\Delta\varepsilon_{cc}$$

$$\sigma_{ab}^{n+1^{trial}} = \sigma_{ab}^{n} + 2G_{ab}\Delta\varepsilon_{ab}$$

$$\sigma_{bc}^{n+1^{trial}} = \sigma_{bc}^{n} + 2G_{bc}\Delta\varepsilon_{bc}$$

$$\sigma_{ca}^{n+1^{trial}} = \sigma_{ca}^{n} + 2G_{ca}\Delta\varepsilon_{ca}$$

Each component of the updated stresses is then independently checked to ensure that they do not exceed the permissible values determined from the load curves; e.g., if

$$\left|\sigma_{ij}^{n+1^{trial}}\right| > \lambda\sigma_{ij}(V)$$

then

$$\sigma_{ij}^{n+1} = \sigma_{ij}(V)\frac{\lambda\sigma_{ij}^{n+1^{trial}}}{\left|\sigma_{ij}^{n+1^{trial}}\right|}$$

On Card 2 σ_{ij}(V) is defined by LCA for the aa stress component, LCB for the bb component, LCC for the cc component, and LCS for the ab, bc, cb shear stress components. The parameter λ is either unity or a value taken from the load curve number, LCSR, that defines λ as a function of strain-rate. Strain-rate is defined here as the Euclidean norm of the deviatoric strain-rate tensor.

For fully compacted material it is assumed that the material behavior is elastic-perfectly plastic and the stress components updated according to:

$$s_{ij}^{trial} = s_{ij}^{n} + 2G\Delta\varepsilon_{ij}^{dev^{n+1/2}}$$

where the deviatoric strain increment is defined as

$$\Delta\varepsilon_{ij}^{dev} = \Delta\varepsilon_{ij} - \frac{1}{3}\Delta\varepsilon_{kk}\delta_{ij} \ .$$

Now a check is made to see if the yield stress for the fully compacted material is exceeded by comparing

$$s_{eff}^{trial} = \left(\frac{3}{2}s_{ij}^{trial}s_{ij}^{trial}\right)^{1/2}$$

the effective trial stress to the defined yield stress, SIGY. If the effective trial stress exceeds the yield stress the stress components are simply scaled back to the yield surface

$$s_{ij}^{n+1} = \frac{\sigma_y}{s_{eff}^{trial}}s_{ij}^{trial} \ .$$

Now the pressure is updated using the elastic bulk modulus, K

$$p^{n+1} = p^{n} - K\Delta\varepsilon_{kk}^{n+1/2}$$

$$K = \frac{E}{3(1-2v)}$$

to obtain the final value for the Cauchy stress

$$\sigma_{ij}^{n+1} = s_{ij}^{n+1} - p^{n+1}\delta_{ij} \ .$$

After completing the stress update transform the stresses back to the global configuration.

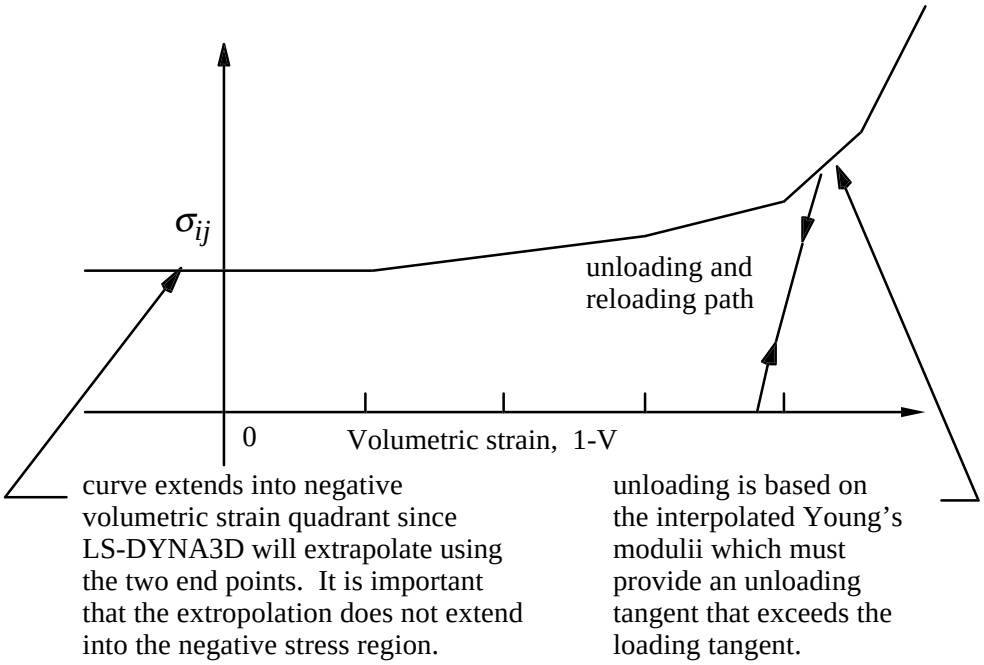

σ_{ij}
unloading and
reloading path
0
Volumetric strain, 1-V
curve extends into negative volumetric strain quadrant since LS-DYNA3D will extrapolate using the two end points. It is important that the extropolation does not extend into the negative stress region.
unloading is based on the interpolated Young's modulii which must provide an unloading tangent that exceeds the loading tangent.

Figure 19.7. Stress quantity versus volumetric strain. Note that the "yield stress" at a volumetric strain of zero is non-zero. In the load curve definition, see *DEFINE_CURVE, the "time" value is the volumetric strain and the "function" value is the yield stress.

***MAT_MOONEY-RIVLIN_RUBBER**

This is Material Type 27. A two-parametric material model for rubber can be defined.

Card Format

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	PR	A	B			
Type	I	F	F	F	F			

Card 2	1	2	3	4	5	6	7	8
Variable	SGL	SW	ST	LCID				
Type	F	F	F	F				

VARIABLE	DESCRIPTION
MID	Material identification. A unique number has to be chosen.
RO	Mass density.
PR	Poisson's ratio (> .49 is recommended, smaller values may not work).
A	Constant, see literature and equations defined below.
B	Constant, see literature and equations defined below.

If A=B=0.0, then a least square fit is computed from tabulated uniaxial data via a load curve. The following information should be defined.

SGL	Specimen gauge length l_0, see Figure 19.8.
SW	Specimen width, see Figure 19.8.
ST	Specimen thickness, see Figure 19.8.
LCID	Load curve ID, see *DEFINE_CURVE, giving the force versus actual change ΔL in the gauge length. See also Figure 19.9 for an alternative definition.

The strain energy density function is defined as:

$$W = A(I-3) + B(II-3) + C(III^{-2} -1) + D (III-1)^2$$

where

$$C = 0.5\ A + B$$

$$D = \frac{A(5\upsilon - 2) + B(11\upsilon - 5)}{2(1 - 2\upsilon)}$$

υ = Poisson's ratio

2(A+B) = shear modulus of linear elasticity

I, II, III = invariants of right Cauchy-Green Tensor $\underset{\sim}{C}$.

The load curve definition that provides the uniaxial data should give the change in gauge length, ΔL, versus the corresponding force. In compression both the force and the change in gauge length must be specified as negative values. In tension the force and change in gauge length should be input as positive values. The principal stretch ratio in the uniaxial direction, λ_1, is then given by

$$\lambda_1 = \frac{L_0 + \Delta L}{L_0}$$

with L_0 being the initial length and L being the actual length.

Alternatively, the stress versus strain curve can also be input by setting the gauge length, thickness, and width to unity (1.0) and defining the engineering strain in place of the change in gauge length and the nominal (engineering) stress in place of the force, see Figure 19.9.

The least square fit to the experimental data is performed during the initialization phase and is a comparison between the fit and the actual input is provided in the printed file. It is a good idea to visually check to make sure it is acceptable. The coefficients A and B are also printed in the output file. It is also advised to use the material driver (see Appendix H) for checking out the material model.

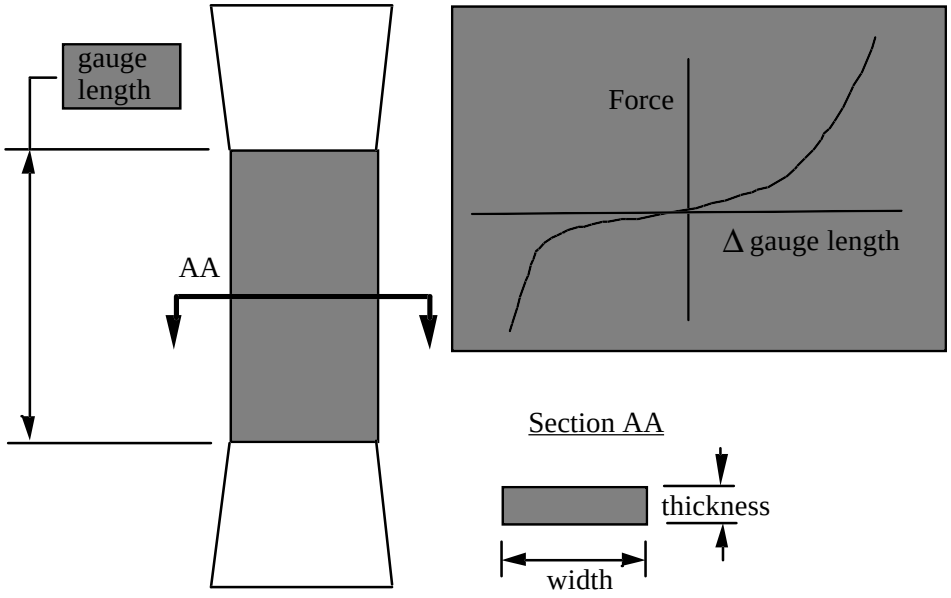


Figure 19.8 Uniaxial specimen for experimental data.

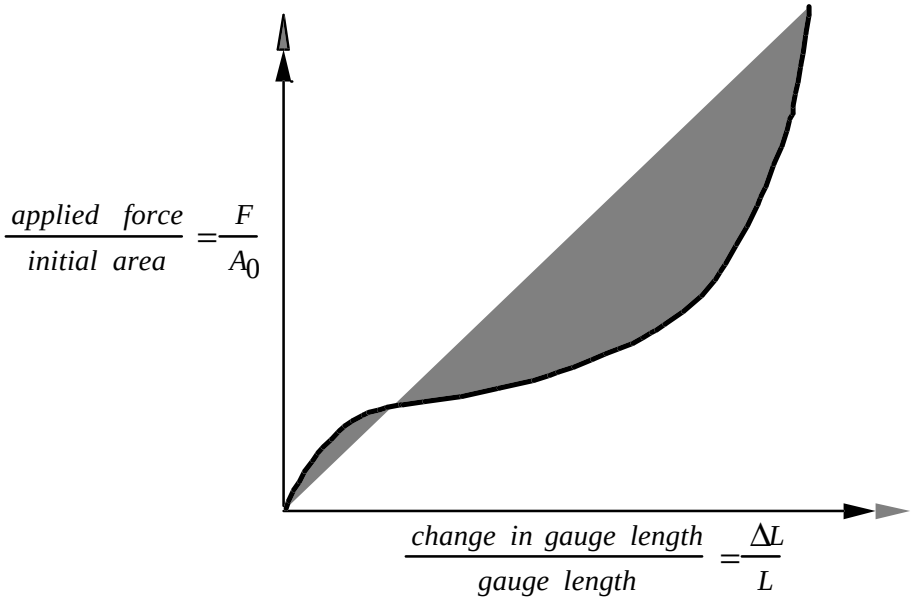


Figure 19.9 The stress versus strain curve can used instead of the force versus the change in the gauge length by setting the gauge length, thickness, and width to unity (1.0) and defining the engineering strain in place of the change in gauge length and the nominal (engineering) stress in place of the force.

*MAT_RESULTANT_PLASTICITY

This is Material Type 28. A resultant formulation for beam and shell elements including elasto-plastic behavior can be defined. This model is available for the Belytschko-Schwer beam, the C^0 triangular shell, and the Belytschko-Tsay shell. For beams, the treatment is elastic-perfectly plastic, but for shell elements isotropic hardening is approximately modeled. For a detailed description we refer to the Theoretical Manual.

Card Format

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	E	PR	SIGY	ETAN		
Type	I	F	F	F	F	F		
Default	none	none	none	none	none	0.0		

VARIABLE	DESCRIPTION
MID	Material identification. A unique number has to be chosen.
RO	Mass density
E	Young's modulus
PR	Poisson's ratio
SIGY	Yield stress
ETAN	Plastic hardening modulus (for shells only)

***MAT_FORCE_LIMITED**

This is Material Type 29. With this material model, for the Belytschko-Schwer beam only, plastic hinge forming at the ends of a beam can be modeled using curve definitions. Optionally, collapse can also be modelled.

Description: FORCE LIMITED Resultant Formulation

Card Format

Card 1

Variable	MID	RO	E	PR	DF	AOPT	YTFLAG	ASOFT
Type	I	F	F	F	F	F	F	F
Default	none	none	none	none	0.0	0.0	0.0	0.0

Card 2

Variable	M1	M2	M3	M4	M5	M6	M7	M8
Type	F	F	F	F	F	F	F	F
Default	none	0	0	0	0	0	0	0

Card 3

Variable	LC1	LC2	LC3	LC4	LC5	LC6	LC7	LC8
Type	F	F	F	F	F	F	F	F
Default	none	0	0	0	0	0	0	0

Card 4

Variable	LPS1	SFS1	LPS2	SFS2	YMS1	YMS2		
Type	F	F	F	F	F	F		
Default	0	1.0	LPS1	1.0	1.0E+20	YMS1		

Card 5

Variable	LPT1	SFT1	LPT2	SFT2	YMT1	YMT2		
Type	F	F	F	F	F	F		
Default	0	1.0	LPT1	1.0	1.0E+20	YMT1		

Card 6

Variable	LPR	SFR	YMR					
Type	F	F	F					
Default	0	1.0	1.0E+20					

VARIABLE	DESCRIPTION
MID	Material identification. A unique number has to be chosen.
RO	Mass density
E	Young's modulus
PR	Poisson's ratio
DF	Damping factor, see definition in notes below. A proper control for the timestep has to be maintained by the user!

VARIABLE	DESCRIPTION
AOPT	Axial load curve option: EQ.0.0: axial load curves are force versus strain, EQ.1.0: axial load curves are force versus change in length .
YTFLAG	Flag to allow beam to yield in tension: EQ.0.0: beam does not yield in tension, EQ.1.0: beam can yield in tension.
ASOFT	Axial elastic softening factor applied once hinge has formed. When a hinge has formed the stiffness is reduced by this factor. If zero, this factor is ignored.
M1, M2,...,M8	Applied end moment for force versus (strain/change in length) curve. At least one must be defined. A maximum of 8 moments can be defined. The values should be in ascending order.
LC1, LC2,...,LC8	Load curve ID (see *DEFINE_CURVE) defining axial force versus strain/change in length (see AOPT) for the corresponding applied end moment. Define the same number as end moments. Each curve must contain the same number of points.
LPS1	Load curve ID for plastic moment versus rotation about s-axis at node 1. If zero, this load curve is ignored.
SFS1	Scale factor for plastic moment versus rotation curve about s-axis at node 1. Default = 1.0.
LPS2	Load curve ID for plastic moment versus rotation about s-axis at node 2. Default: is same as at node 1.
SFS2	Scale factor for plastic moment versus rotation curve about s-axis at node 2. Default: is same as at node 1.
YMS1	Yield moment about s-axis at node 1 for interaction calculations (default set to 1.0E+20 to prevent interaction).
YMS2	Yield moment about s-axis at node 2 for interaction calculations (default set to YMS1).
LPT1	Load curve ID for plastic moment versus rotation about t-axis at node 1. If zero, this load curve is ignored.
SFT1	Scale factor for plastic moment versus rotation curve about t-axis at node 1. Default = 1.0.
LPT2	Load curve ID for plastic moment versus rotation about t-axis at node 2. Default: is the same as at node 1.
SFT2	Scale factor for plastic moment versus rotation curve about t-axis at node 2. Default: is the same as at node 1.

VARIABLE	DESCRIPTION
YMT1	Yield moment about t-axis at node 1 for interaction calculations (default set to 1.0E+20 to prevent interactions)
YMT2	Yield moment about t-axis at node 2 for interaction calculations (default set to YMT1)
LPR	Load curve ID for plastic torsional moment versus rotation. If zero, this load curve is ignored.
SFR	Scale factor for plastic torsional moment versus rotation (default = 1.0).
YMR	Torsional yield moment for interaction calculations (default set to 1.0E+20 to prevent interaction)

This material model is available for the Belytschko resultant beam element only. Plastic hinges form at the ends of the beam when the moment reaches the plastic moment. The moment versus rotation relationship is specified by the user in the form of a load curve and scale factor. The points of the load curve are (plastic rotation in radians, plastic moment). Both quantities should be positive for all points, with the first point being (zero, initial plastic moment). Within this constraint any form of characteristic may be used, including flat or falling curves. Different load curves and scale factors may be specified at each node and about each of the local s and t axes.

Axial collapse occurs when the compressive axial load reaches the collapse load. Collapse load versus collapse deflection is specified in the form of a load curve. The points of the load curve are either (true strain, collapse force) or (change in length, collapse force). Both quantities should be entered as positive for all points, and will be interpreted as compressive. The first point should be (zero, initial collapse load).

The collapse load may vary with end moment as well as with deflections. In this case several load-deflection curves are defined, each corresponding to a different end moment. Each load curve should have the same number of points and the same deflection values. The end moment is defined as the average of the absolute moments at each end of the beam and is always positive.

Stiffness-proportional damping may be added using the damping factor λ. This is defined as follows:

$$\lambda = \frac{2*\xi}{\omega}$$

where ξ is the damping factor at the reference frequency ω (in radians per second). For example if 1% damping at 2Hz is required

$$\lambda = \frac{2*0.01}{2\pi*2} = 0.001592$$

If damping is used, a small timestep may be required. LS-DYNA3D does not check this so to avoid instability it may be necessary to control the timestep via a load curve. As a guide, the timestep required for any given element is multiplied by $0.3L/c\lambda$ when damping is present (L = element length, c = sound speed).

Moment Interaction:

Plastic hinges can form due to the combined action of moments about the three axes. This facility is activated only when yield moments are defined in the material input. A hinge forms when the following condition is first satisfied.

$$\left(\frac{M_r}{M_{ryield}}\right)^2 + \left(\frac{M_s}{M_{syield}}\right)^2 + \left(\frac{M_t}{M_{tyield}}\right)^2 \geq 1$$

where,

M_r, M_s, M_t = current moment
M_{ryield}, M_{syield}, M_{tyield} = yield moment

Note that scale factors for hinge behavior defined in the input will also be applied to the yield moments: for example, M_{syield} in the above formula is given by the input yield moment about the local axis times the input scale factor for the local s axis. For strain-softening characteristics, the yield moment should generally be set equal to the initial peak of the moment-rotation load curve.

On forming a hinge, upper limit moments are set. These are given by

$$M_{r_{upper}} = MAX\left(M_r, \frac{M_{r_{yield}}}{2}\right)$$

and similar for M_s and M_t.

Thereafter the plastic moments will be given by

M_{rp}, = min (M_{rupper}, M_{rcurve}) and similar for s and t

where

M_{rp} = current plastic moment
M_{rcurve} = moment taken from load curve at the current rotation scaled according to the scale factor.

The effect of this is to provide an upper limit to the moment that can be generated; it represents the softening effect of local buckling at a hinge site. Thus if a member is bent about is local s-axis it will then be weaker in torsion and about its local t-axis. For moments-softening curves, the effect is to trim off the initial peak (although if the curves subsequently harden, the final hardening will also be trimmed off).

It is not possible to make the plastic moment vary with axial load.

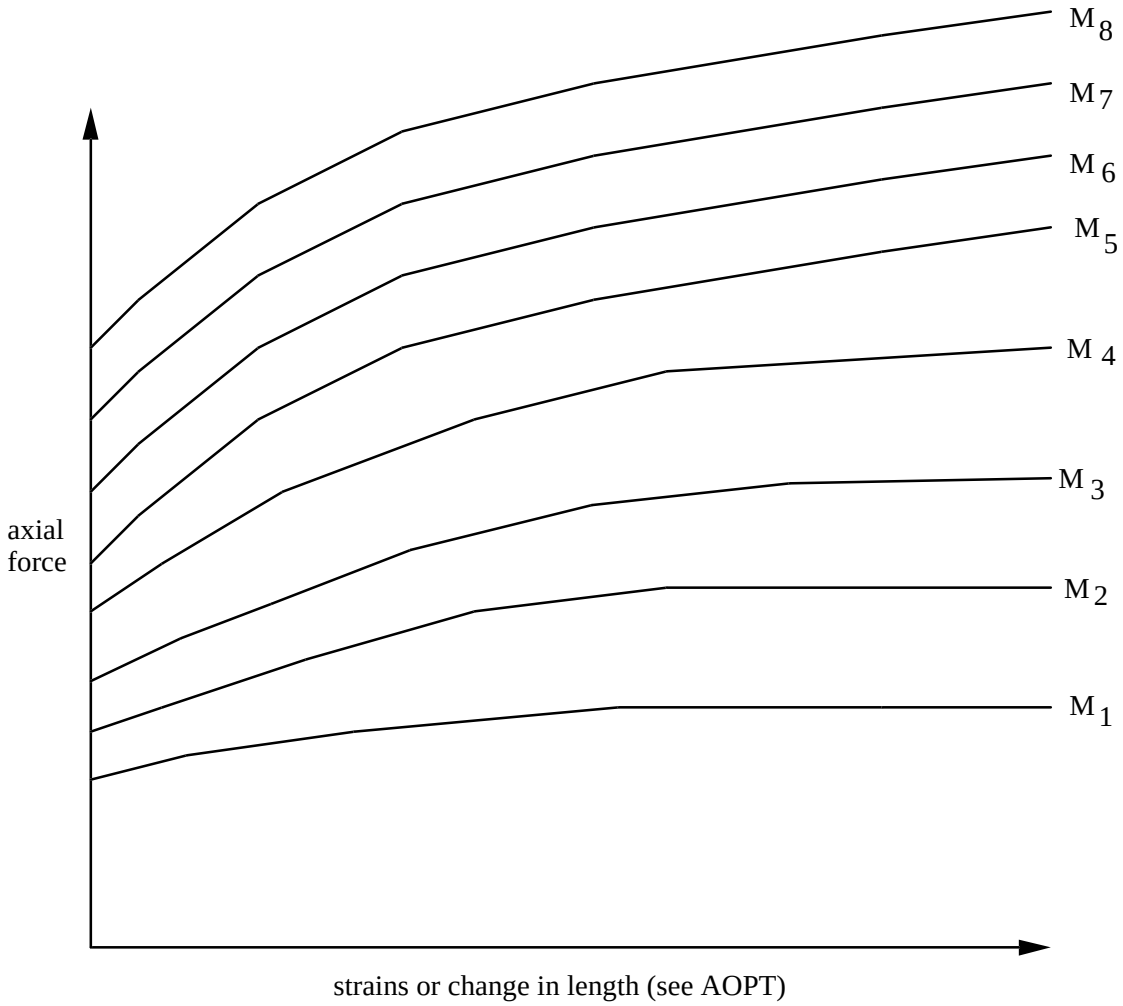


Figure 19.10. The force magnitude is limited by the applied end moment. For an intermediate value of the end moment LS-DYNA3D interpolates between the curves to determine the allowable force value.

*MAT_CLOSED_FORM_SHELL_PLASTICITY

This is Material Type 30. With this model, a non iterative "exact" treatment of the plane stress constitutive equations for elasto-plastic material can be defined. This model is in general more efficient than a fully iterative treatment. However, on vector computers, the vectorized form (i.e., using only 3 iterations) as optionally available in material type 3 is far more efficient. This model is available for perfect plasticity or kinematic hardening for a bi-linear von Mises model. The implementation is described in [Whirley, Hallquist, and Goudreau 1989].

Card Format

	1	2	3	4	5	6	7	8
Variable	MID	RO	E	PR	SIGY	ETAN		
Type	I	F	F	F	F	F		
Default	none	none	none	none	none	0.0		

VARIABLE	DESCRIPTION
MID	Material identification
RO	Density
E	Young's modulus
PR	Poisson's ratio
SIGY	Yield stress
ETAN	Plastic tangent modulus

*MAT_FRAZER_NASH_RUBBER_MODEL

This is Material Type 31. This model defines rubber from uniaxial test data. It is a modified form of the hyperelastic constitutive law first described in [Kendington 1988]. See also the notes below.

Card Format

Card 1

Variable	MID	RO	PR	C100	C200	C300	C400	
Type	I	F	F	F	F	F	F	

Card 2

Variable	C110	C210	C010	C020	EXIT	EMAX	EMIN	
Type	F	F	F	F	F	F	F	

Card 3

Variable	SGL	SW	ST	LCID				
Type	F	F	F	F				

VARIABLE	DESCRIPTION
MID	Material identification.. A unique number has to be defined.
RO	Mass density.
PR	Poisson's ratio. Values between .49 and .50 are suggested.
C100	C_{100} (EQ.1.0 if term is in the least squares fit.).
C200	C_{200} (EQ.1.0 if term is in the least squares fit.).

VARIABLE	DESCRIPTION
C300	C_{300} (EQ.1.0 if term is in the least squares fit.).
C400	C_{400} (EQ.1.0 if term is in the least squares fit.).
C110	C_{110} (EQ.1.0 if term is in the least squares fit.).
C210	C_{210} (EQ.1.0 if term is in the least squares fit.).
C010	C_{010} (EQ.1.0 if term is in the least squares fit.).
C020	C_{020} (EQ.1.0 if term is in the least squares fit.).
EXIT	Exit option: EQ. 0.0: stop if strain limits are exceeded (recommended), NE. 0.0: continue if strain limits are exceeded. The curve is then extrapolated.
EMAX	Maximum strain limit, (Green-St, Venant Strain).
EMIN	Minimum strain limit, (Green-St, Venant Strain).
SGL	Specimen gauge length, see Figure 19.8.
SW	Specimen width, see Figure 19.8.
ST	Specimen thickness, see Figure 19.8.
LCID	Load curve ID, see DEFINE_CURVE, giving the force versus actual change in gauge length. See also Figure 19.9 for an alternative definition.

The constants can be defined directly or a least squares fit can be performed if the uniaxial data (SGL, SW, ST and LCID) is available. If a least squares fit is chosen, then the terms to be included in the energy functional are flagged by setting their corresponding coefficients to unity. If all coefficients are zero the default is to use only the terms involving I_1 and I_2. C_{100} defaults to unity if the least square fit is used.

The strain energy functional, U, is defined in terms of the input constants as:

$$U = C_{100}I_1 + C_{200}I_1^2 + C_{300}I_1^3 + C_{400}I_1^4 + C_{110}I_1I_2 +$$
$$C_{210}I_1^2I_2 + C_{010}I_2 + C_{020}I_2^2 + f(J)$$

where the invariants can be expressed in terms of the deformation gradient matrix, F_{ij}, and the Green-St. Venant strain tensor, E_{ij} :

$$J = \left|F_{ij}\right|$$
$$I_1 = E_{ii}$$
$$I_2 = \frac{1}{2!}\delta^{ij}_{pq}E_{pi}E_{qj}$$

The derivative of U with respect to a component of strain gives the corresponding component of stress

$$S_{ij} = \frac{\partial U}{\partial E_{ij}}$$

here, S_{ij}, is the second Piola-Kirchhoff stress tensor.

The load curve definition that provides the uniaxial data should give the change in gauge length, ΔL, and the corresponding force . In compression both the force and the change in gauge length must be specified as negative values. In tension the force and change in gauge length should be input as positive values. The principal stretch ratio in the uniaxial direction, λ_1, is then given by

$$\lambda_1 = \frac{L_O + \Delta L}{L_O}$$

Alternatively, the stress versus strain curve can also be input by setting the gauge length, thickness, and width to unity and defining the engineering strain in place of the change in gauge length and the nominal (engineering) stress in place of the force, see Figure 19.9.

The least square fit to the experimental data is performed during the initialization phase and is a comparison between the fit and the actual input is provided in the printed file. It is a good idea to visually check the fit to make sure it is acceptable. The coefficients C_{100} - C_{020} are also printed in the output file.

***MAT_LAMINATED_GLASS**

This is Material Type 32. With this material model, a layered glass including polymeric layers can be modeled. Failure of the glass part is possible. See notes below.

Card Format

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	EG	PRG	SYG	ETG	EFG	EP
Type	I	F	F	F	F	F	F	F

Card 2

Variable	PRP	SYP	ETP					
Type	F	F	F					

Card Format. Define 1-4 cards with a maximum of 32 number. If less than 4 cards are input, reading is stopped by a "*" control card.

Card 3, etc.

Variable	F1	F2	F3	F4	F5	F6	F7	F8
Type	F	F	F	F	F	F	F	F

VARIABLE	DESCRIPTION
MID	Material identification. A unique number has to be defined.
RO	Mass density
EG	Young's modulus for glass
PRG	Poisson's ratio for glass

VARIABLE	DESCRIPTION
SYG	Yield stress for glass
ETG	Plastic hardening modulus for glass
EFG	Plastic strain at failure for glass
EP	Young's modulus for polymer
PRP	Poisson's ratio for polymer
SYP	Yield stress for polymer
ETP	Plastic hardening modulus for polymer
F1,..FN	Integration point material: f_n = 0.0: glass, f_n = 1.0: polymer. A user-defined integration rule must be specified, see *INTEGRATION_SHELL.

Isotropic hardening for both materials is assumed. The material to which the glass is bonded is assumed to stretch plastically without failure. A user defined integration rule specifies the thickness of the layers making up the glass. F_i defines whether the integration point is glass (0.0) or polymer (1.0). The material definition, F_i, has to be given for the same number of integration points (NIPTS) as specified in the rule. A maximum of 32 layers is allowed.

*MAT_BARLAT_ANISOTROPIC_PLASTICITY

This is Material Type 33. This model was developed by Barlat, Lege, and Brem [1991] for modelling anisotropic material behavior in forming processes. The finite element implementation of this model is described in detail by Chung and Shah [1992] and is used here. It is based on a six parameter model, which is ideally suited for 3D continuum problems, see notes below. For sheet forming problems, material 35 based on a 3-parameter model is recommended.

Card Format

Card 1

Variable	MID	RO	E	PR	K	E0	N	M
Type	F	F	F	F	F	F	F	F

Card 2

Variable	A	B	C	F	G	H		
Type	F	F	F	F	F	F		

Card 3

Variable	AOPT							
Type	F							

Card 4

Variable	XP	YP	ZP	A1	A2	A3		
Type	F	F	F	F	F	F		

Card 5

Variable	V1	V2	V3	D1	D2	D3		
Type	F	F	F	F	F	F		

VARIABLE	DESCRIPTION
MID	Material identification. A unique number has to be chosen.
RO	Mass density.
E	Young's modulus, E.
PR	Poisson's ratio, ν.
K	k, strength coefficient, see notes below.
EO	ε_0, strain corresponding to the initial yield, see notes below.
N	n, hardening exponent for yield strength.
M	m, flow potential exponent in Barlat's Model.
A	a, anisotropy coefficient in Barlat's Model.
B	b, anisotropy coefficient in Barlat's Model.
C	c anisotropy coefficient in Barlat's Model.
F	f, anisotropy coefficient in Barlat's Model.
G	g, anisotropy coefficient in Barlat's Model.
H	h, anisotropy coefficient in Barlat's Model.
AOPT	Material axes option: EQ. 0.0: locally orthotropic with material axes determined by element nodes as shown in Figure 19.1. Nodes 1, 2, and 4 of an element are identical to the Nodes used for the definition of a coordinate system as by *DEFINE_COORDINATE_NODES.

VARIABLE	DESCRIPTION
	EQ. 1.0: locally orthotropic with material axes determined by a point in space and the global location of the element center, this is the a-direction. EQ. 2.0: globally orthotropic with material axes determined by vectors defined below, as with *DEFINE_COORDINATE_VECTOR. EQ. 3.0: applicable to shell elements only. This option determines locally orthotropic material axes by offsetting the material axes by an angle to be specified from a line in the plane of the shell determined by taking the cross product of the vector **v** defined below with the shell normal vector.
XP YP ZP	Coordinates of point p for AOPT = 1.
A1 A2 A3	Components of vector a for AOPT = 2.
V1 V2 V3	Components of vector v for AOPT = 3.
D1 D2 D3	Components of vector d for AOPT = 2.

The yield function Φ is defined as

$$\Phi = |S_1-S_2|^m + |S_2-S_3|^m + |S_3-S_1|^m = 2\,\bar{\sigma}^{\,m}$$

where $\bar{\sigma}$ is the effective stress and $S_{i=1,2,3}$ are the principal values of the symmetric matrix $S_{\alpha\beta}$,

$$\begin{aligned}
S_{xx} &= [c(\sigma_{xx} - \sigma_{yy}) - b(\sigma_{zz} - \sigma_{xx})] / 3 \\
S_{yy} &= [a(\sigma_{yy} - \sigma_{zz}) - c(\sigma_{xx} - \sigma_{yy})] / 3 \\
S_{zz} &= [b(\sigma_{zz} - \sigma_{xx}) - a(\sigma_{yy} - \sigma_{zz})] / 3 \\
S_{yz} &= f\,\sigma_{yz} \\
S_{zx} &= g\,\sigma_{zx} \\
S_{xy} &= h\,\sigma_{xy}
\end{aligned}$$

The material constants a, b, c, f, g and h represent anisotropic properties. When a=b=c=f=g=h=1, the material is isotropic and the yield surface reduces to the Tresca yield surface for m=1 and von Mises yield surface for m= 2 or 4.

For face centered cubic (FCC) materials m=8 is recommended and for body centered cubic (BCC) materials m=6 is used. The yield strength of the material is

$$\sigma_y = k\left(\varepsilon^p + \varepsilon_0\right)^n$$

where ε_0 is the strain corresponding to the initial yield stress and ε^p is the plastic strain.

*MAT_FABRIC

This is Material Type 34. This material is especially developed for airbag materials. The fabric model is a variation on the layered orthotropic composite model of material 22 and is valid for 3 and 4 node membrane elements only. In addition to being a constitutive model, this model also invokes a special membrane element formulation which is more suited to the deformation experienced by fabrics under large deformation. For thin fabrics, buckling can result in an inability to support compressive stresses; thus a flag is included for this option. A linearly elastic liner is also included which can be used to reduce the tendency for these elements to be crushed when the no-compression option is invoked. In LS-DYNA3D versions after 931 the isotropic elastic option is available.

Card Format

Card 1

Variable	MID	RO	EA	EB	EC	PRBA	PRCA	PRCB
Type	F	F	F	F	F	F	F	F

Card 2

Variable	GAB	GBC	GCA	CSE	EL	PRL	LRATIO	DAMP
Type	F	F	F	F	F	F	F	F
Remarks				1	2	2	2	

Card 3

Variable	AOPT							
Type	F							

Card 4

Variable	XP	YP	ZP	A1	A2	A3		
Type	F	F	F	F	F	F		

Card 5

Variable	V1	V2	V3	D1	D2	D3		
Type	F	F	F	F	F	F		

VARIABLE	DESCRIPTION
MID	Material identification. A unique number has to be chosen.
RO	Mass density.
EA	Young's modulus - longitudinal direction. For an isotopic elastic fabric material only EA and PRBA are defined and are used as the isotropic Young's modulus and Poisson's ratio, respectively. The input for the fiber directions and liner should be input as zero for the isotropic elastic fabric.
EB	Young's modulus - transverse direction, set to zero for isotropic elastic material.
EC	Young's modulus - normal direction, set to zero for isotropic elastic material.
PRBA	ν_{ba}, Poisson's ratio ba direction.
PRCA	ν_{ca}, Poisson's ratio ca direction, set to zero for isotropic elastic material.
PRCB	ν_{cb}, Poisson's ratio cb direction, set to zero for isotropic elastic material.
GAB	G_{ab}, shear modulus ab direction, set to zero for isotropic elastic material.
GBC	G_{bc}, shear modulus bc direction, set to zero for isotropic elastic material.
GCA	G_{ca}, shear modulus ca direction, set to zero for isotropic elastic material.

VARIABLE	DESCRIPTION
CSE	Compressive stress elimination option (default 0.0): EQ.0.0: don't eliminate compressive stresses, EQ.1.0: eliminate compressive stresses.
EL	Young's modulus for elastic liner (optional).
PRL	Poisson's ratio for elastic liner (optional).
LRATIO	Ratio of liner thickness to total fabric thickness.
DAMP	Rayleigh damping coefficient. A 0.05 coefficient is recommended corresponding to 5% of critical damping. Sometimes larger values are necessary.
AOPT	Material axes option: EQ. 0.0: locally orthotropic with material axes determined by element nodes as shown in Figure 19.1. Nodes 1, 2, and 4 of an element are identical to the Nodes used for the definition of a coordinate system as by *DEFINE_COORDINATE_NODES. EQ. 1.0: locally orthotropic with material axes determined by a point in space and the global location of the element center, this is the a-direction. EQ. 2.0: globally orthotropic with material axes determined by vectors defined below, as with *DEFINE_COORDINATE_VECTOR. EQ. 3.0: applicable to shell elements only. This option determines locally orthotropic material axes by offsetting the material axes by an angle to be specified from a line in the plane of the shell determined by taking the cross product of the vector **v** defined below with the shell normal vector.
XP YP ZP	Coordinates of point p for AOPT = 1.
A1 A2 A3	Components of vector a for AOPT = 2.
V1 V2 V3	Components of vector v for AOPT = 3.
D1 D2 D3	Components of vector d for AOPT = 2.

Remarks:

1. The no compression option allows the simulation of airbag inflation with far less elements than would be needed for the discritization of the wrinkles which would occur for the case when compressive stresses are not eliminated.

2. When using this material for the analysis of membranes as airbags it is well known from classical theory that only one layer has to be defined. The so-called elastic liner has to be defined for numerical purposes only when the no compression option is invoked.

*MAT_PLASTIC_GREEN-NAGHDI_RATE

This is Material Type 35. This model is available only for brick elements and is similar to model 3, but uses the Green-Naghdi Rate formulation rather than the Jaumann rate for the stress update. For some cases this might be helpful. This model also has a strain rate dependency following the Cowper-Symonds model.

Card Format

Card 1

Variable	MID	RO	E	PR				
Type	I	F	F	F				

Card 2

Variable	SIGY	ETAN	SRC	SRP	BETA			
Type	F	F	F	F	F			

VARIABLE	DESCRIPTION
MID	Material identification
RO	Density
E	Young's modulus
PR	Poisson's ratio
SIGY	Yield stress
ETAN	Plastic hardening modulus
SRC	Strain rate parameter, C
SRP	Strain rate parameter, P
BETA	Hardening parameter, $0 < \beta' < 1$

*MAT_3-PARAMETER_BARLAT

This is Material Type 36. This model was developed by Barlat and Lian [1989] for modelling sheets with anisotropic materials under plane stress conditions. This material allows the use of the Lankford parameters for the definition of the anisotropy. This particular development is due to Barlat and Lian [1989].

Card Format

Card 1

Variable	MID	RO	E	PR	HR	P1	P2	
Type	F	F	F	F	F	F	F	

Card 2

Variable	M	R00	R45	R90				
Type	F	F	F	F				

Card 3

Variable	AOPT							
Type	F							

Card 4

Variable	XP	YP	ZP	A1	A2	A3		
Type	F	F	F	F	F	F		

Card 5

Variable	V1	V2	V3	D1	D2	D3		
Type	F	F	F	F	F	F		

VARIABLE	DESCRIPTION
MID	Material identification. A unique number has to be chosen.
RO	Mass density.
E	Young's modulus, E
PR	Poisson's ratio, ν
HR	Hardening rule: EQ.1.0: linear (default), EQ.2.0: exponential.
P1	Material parameter: HR.EQ.1.0: Tangent modulus, HR.EQ.2.0: k, strength coefficient for exponential hardening
P2	Material parameter: HR.EQ.1.0: Yield stress HR.EQ.2.0: n, exponent
M	m, exponent in Barlat's yield surface
R00	R_{00}, Lankford parmeter determined from experiments
R45	R_{45}, Lankford parmeter determined from experiments
R90	R_{90}, Lankford parmeter determined from experiments
AOPT	Material axes option: EQ. 0.0: locally orthotropic with material axes determined by element nodes as shown in Figure 19.1. Nodes 1, 2, and 4 of an element are identical to the Nodes used for the definition of a coordinate system as by *DEFINE_COORDINATE_NODES. EQ. 1.0: locally orthotropic with material axes determined by a point in space and the global location of the element center, this is the a-direction. EQ. 2.0: globally orthotropic with material axes determined by vectors defined below, as with *DEFINE_COORDINATE_VECTOR.

VARIABLE	DESCRIPTION
	EQ. 3.0: applicable to shell elements only. This option determines locally orthotropic material axes by offsetting the material axes by an angle to be specified from a line in the plane of the shell determined by taking the cross product of the vector **v** defined below with the shell normal vector.
XP YP ZP	Coordinates of point p for AOPT = 1.
A1 A2 A3	Components of vector a for AOPT = 2.
V1 V2 V3	Components of vector v for AOPT = 3.
D1 D2 D3	Components of vector d for AOPT = 2.

The anisotopic yield criterion Φ for plane stress is defined as:

$$\Phi = a\left|K_1 + K_2\right|^m + a\left|K_1 - K_2\right|^m + c\left|2K_2\right|^m = 2\sigma_Y^m$$

where σ_Y is the yield stress and $K_{i=1,2}$ are given by:

$$K_1 = \frac{\sigma_x - h\sigma_y}{2}$$

$$K_2 = \sqrt{\left(\frac{\sigma_x - h\sigma_y}{2}\right)^2 + p^2\tau_{xy}^2}$$

The anisotropic material constants a, c, h, and p are obtained through R_{00}, R_{45}, and R_{90}:

$$a = 2 - 2\sqrt{\frac{R_{00}}{1+R_{00}}\frac{R_{90}}{1+R_{90}}}$$

$$c = 2 - a$$

$$h = \sqrt{\frac{R_{00}}{1+R_{00}}\frac{1+R_{90}}{R_{90}}}$$

The anisotropy parameter p is calculated implicitly. According to Barlat and Lian the R value, width to thickness strain ratio, for any angle ϕ can be calculated from:

$$R_\phi = \frac{2m\sigma_Y^m}{\left(\frac{\partial\Phi}{\partial\sigma_x} + \frac{\partial\Phi}{\partial\sigma_y}\right)\sigma_\phi} - 1$$

where σ_ϕ is the uniaxial tension in the ϕ direction. This expression can be used to iteratively calculate the value of p. Let ϕ=45 and define a function g as

$$g(p) = \frac{2m\sigma_Y^m}{\left(\frac{\partial\Phi}{\partial\sigma_x} + \frac{\partial\Phi}{\partial\sigma_y}\right)\sigma_\phi} - 1 - R_{45}$$

An iterative search is used to find the value of p.

For face centered cubic (FCC) materials m=8 is recommended and for body centered cubic (BCC) materials m=6 may be used. The yield strength of the material can be expressed in terms of k and n:

$$\sigma_Y = k\left(\varepsilon^p + \varepsilon_0\right)^n$$

where ε_0 is the strain corresponding to the initial yield stress and ε^p is the plastic strain.

*MAT_TRANSVERSELY_ANISOTROPIC_ELASTIC_PLASTIC

This is Material Type 37. This model is for simulating sheet forming processes with anisotropic material. Only transverse anisotropy can be considered. Optionally an arbitrary dependency of stress and effective plastic strain can be defined via a load curve. This plasticity model is fully iterative and is available only for shell elements. Also see the notes below.

Card Format

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	E	PR	SIGY	ETAN	R	HLCID
Type	I	F	F	F	F	F	F	F

VARIABLE	DESCRIPTION
MID	Material identification. A unique number has to be chosen.
RO	Mass density.
E	Young's modulus.
PR	Poisson's ratio.
SIGY	Yield stress.
ETAN	Plastic hardening modulus.
R	Anisotropic hardening parameter.
HLCID	Load curve ID defining effective yield stress versus effective plastic strain.

Consider Cartesian reference axes which are parallel to the three symmetry planes of anisotropic behavior. Then, the yield function suggested by [Hill 1948] can be written

$$F(\sigma_{22}-\sigma_{33})^2 + G(\sigma_{33}-\sigma_{11})^2 + H(\sigma_{11}-\sigma_{22})^2 + 2L\sigma_{23}^2 + 2M\sigma_{31}^2 + 2N\sigma_{12}^2 - 1 = 0$$

where σ_{y1}, σ_{y2}, and σ_{y3}, are the tensile yield stresses and σ_{y12}, σ_{y23}, and σ_{y31} are the shear yield stresses. The constants F, G H, L, M, and N are related to the yield stress by

$$2L = \frac{1}{\sigma_{23}^2}$$

$$2M = \frac{1}{\sigma_{y31}^2}$$

$$2N = \frac{1}{\sigma_{y12}^2}$$

$$2F = \frac{1}{\sigma_{y2}^2} + \frac{1}{\sigma_{y3}^2} - \frac{1}{\sigma_{y1}^2}$$

$$2G = \frac{1}{\sigma_{y3}^2} + \frac{1}{\sigma_{y1}^2} - \frac{1}{\sigma_{y2}^2}$$

$$2H = \frac{1}{\sigma_{y1}^2} + \frac{1}{\sigma_{y2}^2} - \frac{1}{\sigma_{y3}^2}$$

The isotropic case of von Mises plasticity can be recovered by setting F = G = H = $\frac{1}{2\sigma_y^2}$

and $$L = M = N = \frac{3}{2\sigma_y^2} \ .$$

For the particular case of transverse anisotropy, where properties do not vary in the x_1-x_2 plane, the following relations hold:

$$2F = 2G = \frac{1}{\sigma_{y3}^2}$$

$$2H = \frac{2}{\sigma_y^2} - \frac{1}{\sigma_{y3}^2}$$

$$N = \frac{2}{\sigma_y^2} - \frac{1}{2}\frac{1}{\sigma_{y3}^2}$$

where it has been assumed that $\sigma_{y1} = \sigma_{y2} = \sigma_y$.

Letting $K = \frac{\sigma_y}{\sigma_{y3}}$, the yield criteria can be written

$$F(\boldsymbol{\sigma}) = \sigma_e = \sigma_y \ ,$$

where

$$F(\sigma) \equiv \left[\sigma_{11}^2 + \sigma_{22}^2 + K^2\sigma_{33}^2 - K^2\sigma_{33}(\sigma_{11}+\sigma_{22}) - \left(2-K^2\right)\sigma_{11}\sigma_{22} \right.$$
$$\left. + 2L\sigma_y^2\left(\sigma_{23}^2+\sigma_{31}^2\right) + 2\left(2-\frac{1}{2}K^2\right)\sigma_{11}^2\right]^{1/2}$$

The rate of plastic strain is assumed to be normal to the yield surface so $\dot{\varepsilon}_{ij}^p$ is found from

$$\dot{\varepsilon}_{ij}^p = \lambda\frac{\partial F}{\partial\sigma_{ij}} \ .$$

Now consider the case of plane stress, where $\sigma_{33} = 0$. Also, define the anisotropy input parameter, R, as the ratio of the in-plane plastic strain rate to the out-of-plane plastic strain rate,

$$R = \frac{\dot{\varepsilon}_{22}^p}{\dot{\varepsilon}_{33}^p} \ .$$

It then follows that

$$R = \frac{2}{K^2} - 1 \ .$$

Using the plane stress assumption and the definition of R, the yield function may now be written

$$F(\sigma) = \left[\sigma_{11}^2 + \sigma_{22}^2 - \frac{2R}{R+1}\sigma_{11}\sigma_{22} + 2\frac{2R+1}{R+1}\sigma_{12}^2\right]^{1/2} \ .$$

*MAT_BLATZ-KO_FOAM

This is Material Type 38. This model is for the definition of rubber like foams of polyurethane. It is a simpe one-parameter model with a fixed Poisson's ratio of .25.

Card Format

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	G					
Type	I	F	F					

VARIABLE	DESCRIPTION
MID	Material identification. A unique number has to be chosen.
RO	Mass density.
G	Shear modulus.

The strain energy functional for the compressible foam model is given by

$$W = \frac{G}{2}\left(\frac{\mathrm{II}}{\mathrm{III}} + 2\sqrt{\mathrm{III}} - 5\right)$$

Blatz and Ko [1962] suggested this form for a 47 percent volume polyurethane foam rubber with a Poisson's ratio of 0.25. In terms of the strain invariants, I, II, and III, the second Piola-Kirchhoff stresses are given as

$$S^{ij} = G\left[\left(I\delta_{ij} - C_{ij}\right)\frac{1}{\mathrm{III}} + \left(\sqrt{\mathrm{III}} - \frac{\mathrm{II}}{\mathrm{III}}\right)C_{ij}^{-1}\right]$$

where C_{ij} is the right Cauchy-Green strain tensor. This stress measure is transformed to the Cauchy stress, σ_{ij}, according to the relationship

$$\sigma^{ij} = \mathrm{III}^{-1/2} F_{ik} F_{jl} S_{lk}$$

where F_{ij} is the deformation gradient tensor.

The second Piola-Kirchhoff stress is computed as

$$S_{ij} = \mu\left(\frac{1}{V}C_{ij} - V^{-1/1-2\upsilon}\delta_{ij}\right)$$

where V is the relative volume, C_{ij} is the right Cauchy-Green strain tensor, and ν is Poisson's ratio, which is set to .25 internally. This stress measure is transformed to the Cauchy stress, $\sigma_{ij,}$ according to the relationship

$$\sigma_{ij} = V^{-1} F_{ik} F_{jl} S_{lk}$$

where F_{ij} is the deformation gradient tensor.

*MAT_FLD_TRANSVERSELY_ANISOTROPIC

This is Material Type 39. This model is for simulating sheet forming processes with anisotropic material. Only transverse anisotropy can be considered. Optionally, an arbitrary dependency of stress and effective plastic strain can be defined via a load curve. A flow limit diagram can be defined using a curve and is used to compute the maximum strain ratio which can be plotted in LS-TAURUS. This plasticity model is fully iterative and is available only for shell elements. Also see the notes below.

Card Format

Card 1

Variable	MID	RO	E	PR	SIGY	ETAN	R	HLCID
Type	I	F	F	F	F	F	F	F

Card 2

Variable	LCIDFLD							
Type	F							

VARIABLE	DESCRIPTION
MID	Material identification. A unique number has to be chosen.
RO	Mass density.
E	Young's modulus.
PR	Poisson's ratio.
SIGY	Yield stress.
ETAN	Plastic hardening modulus, see notes for model 37.
R	Anisotropic hardening parameter, see notes for model 37.

VARIABLE	DESCRIPTION
HLCID	Load curve ID defining effective stress versus effective plastic strain. The yield stress and hardening modulus are ignored with this option.
LCIDFLD	Load curve ID defining the Flow Limit Diagram. Minor strains in percent are defined as abcissa values and Major strains in percent are defined as ordinate values. The flow limit diagram is shown in Figure 19.11. In defining the curve list pairs of minor and major strains starting with the left most point and ending with the right most point, see *DEFINE_CURVE.

See material model 37 for the theoretical basis. The first history variable is the maximum strain ratio defined by:

$$\frac{\varepsilon_{major_{workpiece}}}{\varepsilon_{major_{fld}}}$$

corresponding to $\varepsilon_{minor_{workpiece}}$.

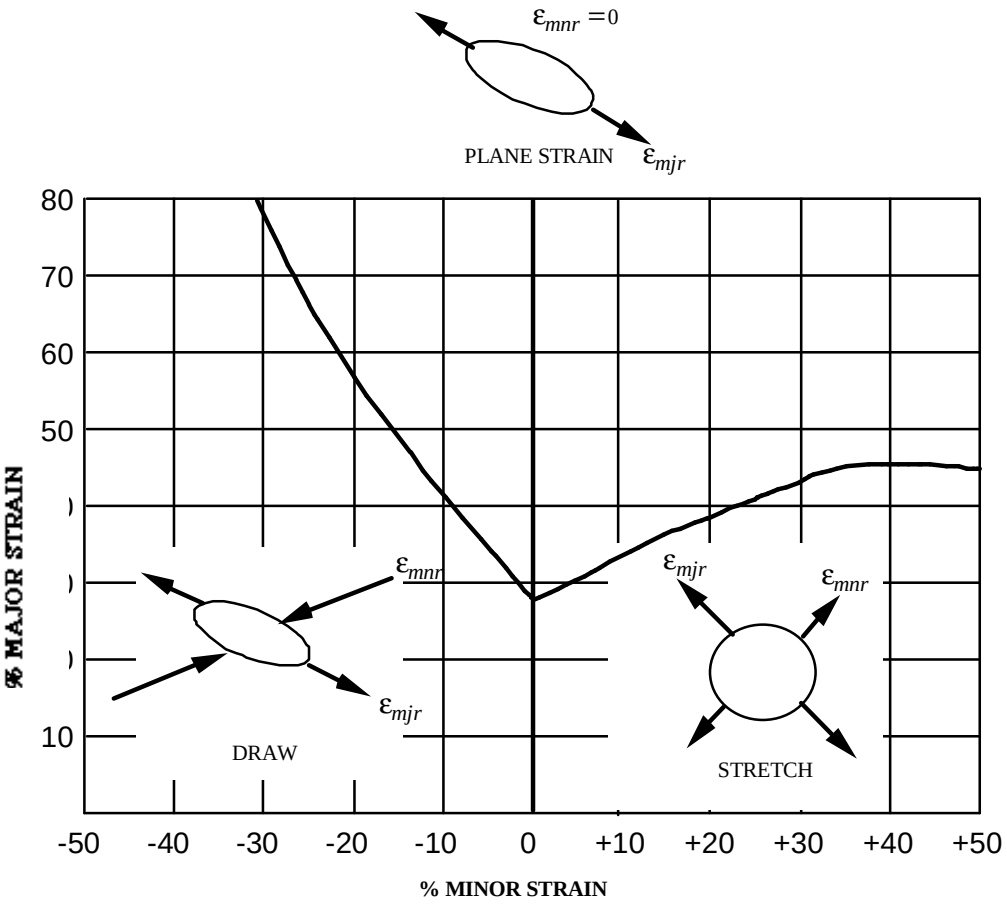


Figure 19.11. Flow limit diagram.

*MAT_NONLINEAR_ORTHOTROPIC

This is Material Type 40. This model allows the definition of an orthotropic nonlinear elastic material based on a finite strain formulation with the initial geometry as the reference. Failure is optional with two failure criteria available. Optionally, stiffness proportional damping can be defined. In the stress initialization phase, temperatures can be varied to impose the initial stresses. This model is only available for shell elements.

Card Format

Card 1

Variable	MID	RO	EA	EB	EC	PRBA	PRCA	PRCB
Type	I	F	F	F	F	F	F	F
Default	none	none	none	none	none	none	none	none

Card 2

Variable	GAB	GBC	GCA	DT	TRAMP	ALPHA		
Type	F	F	F	F	F	F		
Default	none	none	none	0	0	0		

Card 3

Variable	LCIDA	LCIDB	EFAIL	DTFAIL	CDAMP	AOPT		
Type	F	F	F	F	F	F		
Default	0.0	0.0	0.0	0.0	0.0	0.0		

Card 4	1	2	3	4	5	6	7	8
Variable	XP	YP	ZP	A1	A2	A3		
Type	F	F	F	F	F	F		

Card 5	1	2	3	4	5	6	7	8
Variable	V1	V2	V3	D1	D2	D3		
Type	F	F	F	F	F	F		

VARIABLE	DESCRIPTION
MID	Material identification. A unique number has to be chosen.
RO	Mass density.
EA	E_a, Young's modulus in a-direction.
EB	E_b, Young's modulus in b-direction.
EC	E_c, Young's modulus in c-direction.
PRBA	ν_{ba}, Poisson's ratio ba.
PRCA	ν_{ca}, Poisson's ratio ca.
PRCB	ν_{cb}, Poisson's ratio cb.
GAB	G_{ab}, shear modulus ab.
GBC	G_{bc}, shear modulus bc.
GCA	G_{ca}, shear modulus ca.
DT	Temperature increment for isotropic stress initialization. This option can be used during dynamic relaxation.
TRAMP	Time to ramp up to the final temperature.
ALPHA	Thermal expansion coefficient.

VARIABLE	DESCRIPTION
LCIDA	Optional load curve ID defining the nominal stress versus strain along a-axis. Strain is defined as λ_a-1 where λ_a is the stretch ratio along the a axis.
LCIDB	Optional load curve ID defining the nominal stress versus strain along b-axis. Strain is defined as λ_b-1 where λ_b is the stretch ratio along the b axis.
EFAIL	Failure strain, λ-1.
DTFAIL	Time step for automatic element erosion
CDAMP	Damping coefficient.
AOPT	Material axes option: EQ. 0.0: locally orthotropic with material axes determined by element nodes as shown in Figure 19.1. Nodes 1, 2, and 4 of an element are identical to the Nodes used for the definition of a coordinate system as by *DEFINE_COORDINATE_NODES. EQ. 1.0: locally orthotropic with material axes determined by a point in space and the global location of the element center, this is the a-direction. EQ. 2.0: globally orthotropic with material axes determined by vectors defined below, as with *DEFINE_COORDINATE_VECTOR. EQ. 3.0: applicable to shell elements only. This option determines locally orthotropic material axes by offsetting the material axes by an angle to be specified from a line in the plane of the shell determined by taking the cross product of the vector **v** defined below with the shell normal vector.
XP,YP,ZP	x_p y_p z_p, define coordinates of point **p** for AOPT = 1
A1,A2,A3	a_1 a_2 a_3, define components of vector **a** for AOPT = 2.
D1,D2,D3	d_1 d_2 d_3, define components of vector **d** for AOPT = 2.
V1,V2,V3	v_1 v_2 v_3, define components of vector **v** for AOPT = 3

***MAT_USER_DEFINED_MATERIAL_MODELS**

These are Material Types 41-50. The user can supply his own subroutines. See also Appendix A. The keyword input has to be used for the user interface with data. Isotopic and anisotropic material models with failure can be handled.

Card Format

Card 1

Variable	MID	RO	MT	LMC	NHV	IORTHO	IBULK	IG
Type	I	F	I	I	I	I	I	I

Card 2

Variable	IVECT	IFAIL						
Type	I	I						

Define the following two cards if and only if IORTHO=1

Card 3

Variable	AOPT	MAXC	XP	YP	ZP	A1	A2	A3
Type	F	F	F	F	F	F	F	F

Card 4

Variable	V1	V2	V3	D1	D2	D3		
Type	F	F	F	F	F	F		

Define LMC material parameters using 8 parameters per card.

Card	1	2	3	4	5	6	7	8
Variable	P1	P2	P3	P4	P5	P6	P7	P8
Type	F	F	F	F	F	F	F	F

VARIABLE	DESCRIPTION
MID	Material identification. A unique number has to be chosen.
RO	Mass density.
MT	User material type (41-50 inclusive). A number between 41 and 50 has to be chosen.
LMC	Length of material constant array which is equal to the number of material constants to be input.
NHV	Number of history variables to be stored, see Appendix A.
IORTHO	Set to 1 if the material is orthotropic.
IBULK	Address of bulk modulus in material constants array, see Appendix A.
IG	Address of shear modulus in material constants array, see Appendix A.
IVECT	Vectorization flag (on=1). A vectorized user subroutine must be supplied.
IFAIL	Failure flag (on=1). Allows failure of the elements due to a material failure criterion.
AOPT	Material axes option: EQ. 0.0: locally orthotropic with material axes determined by element nodes as shown in Figure 19.1. Nodes 1, 2, and 4 of an element are identical to the Nodes used for the definition of a coordinate system as by *DEFINE_COORDINATE_NODES, EQ. 1.0: locally orthotropic with material axes determined by a point in space and the global location of the element center, this is the a-direction, EQ. 2.0: globally orthotropic with material axes determined by vectors defined below, as with *DEFINE_COORDINATE_VECTOR, EQ. 3.0: applicable to shell elements only. This option determines locally orthotropic material axes by offsetting the material axes by an angle to be specified from a line in the plane of the shell determined by taking the cross product of the vector **v** defined below with the shell normal vector.

VARIABLE	DESCRIPTION
MAXC	Material axes change flag for brick elements for quick changes: EQ.1.0: default, EQ.2.0: switch material axes a and b, EQ.3.0: switch material axes a and c.
XP YP ZP	Coordinates of point p for AOPT = 1.
A1 A2 A3	Components of vector a for AOPT = 2.
V1 V2 V3	Components of vector v for AOPT = 3.
D1 D2 D3	Components of vector d for AOPT = 2.
P1	First material parameter.
P2	Second material parameter.
P3	Third material parameter.
P4	Fourth material parameter.
.	.
.	.
.	.
PLCM	LCMth material parameter.

*MAT_BAMMAN

This is Material Type 51. It allows the modeling of temperature and rate dependent plasticity with a fairly complex model that has many input parameters [Bamman, 1989].

Card Format

Card 1

Variable	MID	RO	E	PR	T	HC		
Type	I	F	F	F	F	F		

Card 2

Variable	C1	C2	C3	C4	C5	C6	C7	C8
Type	F	F	F	F	F	F	F	F

Card 3

Variable	C9	C10	C11	C12	C13	C14	C15	C16
Type	F	F	F	F	F	F	F	F

Card 4

Variable	C17	C18	A1	A2	A3	A4	A5	A6
Type	F	F	F	F	F	F	F	F

VARIABLE	DESCRIPTION
MID	Material identification. A unique number has to be chosen.
RO	Mass density.
E	Young's modulus (psi)
PR	Poisson's ratio
T	Initial temperature (^{o}R)
HC	Heat generation coefficient ($^{oR}/_{psi}$)
C1	Psi
C2	^{o}R
C3	Psi
C4	^{o}R
C5	$^{1}/_{s}$
C6	^{o}R
C7	$^{1}/_{psi}$
C8	^{o}R
C9	Psi
C10	^{o}R
C11	$^{1}/_{psi\text{-}s}$
C12	^{o}R
C13	$^{1}/_{psi}$
C14	^{o}R
C15	psi
C16	^{o}R
C17	$^{1}/_{psi\text{-}s}$
C18	^{o}R

VARIABLE	DESCRIPTION
A1	α_1, initial value of internal state variable 1
A2	α_2, initial value of internal state variable 2
A3	α_4, initial value of internal state variable 3
A4	α_5, initial value of internal state variable 4
A5	α_6, initial value of internal state variable 5
A6	κ, initial value of internal state variable 6

sec-psi-^{o}R	sec-MPa-^{o}R	sec-MPA-^{o}K
C_1	$*1/145$	$*1/145$
C_2	—	$*5/9$
C_3	$*1/145$	$*1/145$
C_4	—	$*5/9$
C_5	—	—
C_6	—	$*5/9$
C_7	*145	*145
C_8	—	$*5/9$
C_9	$*1/145$	$*1/145$
C_{10}	—	$*5/9$
C_{11}	*145	*145
C_{12}	—	$*5/9$
C_{13}	*145	*145
C_{14}	—	$*5/9$
C_{15}	$*1/145$	$*1/145$
C_{16}	—	$*5/9$
C_{17}	*145	*145
C_{18}	—	$*5/9$
C0=HC	*145	$*145*5/9$
E	$*1/145$	$*1/145$
υ	—	—
T	—	$*5/9$

The kinematics associated with the model are discussed in references [Hill 1948, Bammann and Aifantis 1987, Bammann 1989]. The description below is taken nearly verbatim from Bammann [1989].

With the assumption of linear elasticity we can write,

$$\overset{o}{\sigma} = \lambda tr\left(D^e\right)1 + 2D^e$$

where the Cauchy stress σ is convected with the elastic spin W^e as,

$$\overset{o}{\sigma} = \dot{\sigma} - W^e\sigma + \sigma W^e$$

This is equivalent to writing the constitutive model with respect to a set of directors whose direction is defined by the plastic deformation [Bammann and Aifantis 1987, Bammann and Johnson 1987]. Decomposing both the skew symmetric and symmetric parts of the velocity gradient into elastic and plastic parts we write for the elastic stretching D^e and the elastic spin W^e,

$$D^e = D - D^p - D^{th}, \quad W^e = W = W^p .$$

Within this structure it is now necessary to prescribe an equation for the plastic spin W^p in addition to the normally prescribed flow rule for D^p and the stretching due to the thermal expansion D^{th}. As proposed, we assume a flow rule of the form,

$$D^p = f(T)\sinh\left[\frac{|\xi| - \kappa - Y(T)}{V(T)}\right]\frac{\xi'}{|\xi'|} .$$

where T is the temperature, κ is the scalar hardening variable, and ξ' is the difference between the deviatoric Cauchy stress σ' and the tensor variable α',

$$\xi' = \sigma' - \alpha'$$

and f(T), Y(T), V(T) are scalar functions whose specific dependence upon the temperature is given below. Assuming isotropic thermal expansion and introducing the expansion coefficient $\dot{A}$, the thermal stretching can be written,

$$D^{th} = \dot{A}\dot{T}1 .$$

The evolution of the internal variables α and κ are prescribed in a hardening minus recovery format as,

$$\overset{o}{\alpha} = h(T)D^p - \left[r_d(T)\left|D^p\right| + r_s(T)\right]|\alpha|\alpha \ ,$$

$$\dot{\kappa} = H(T)D^p - \left[R_d(T)\left|D^p\right| - R_s(T)\right]\kappa^2$$

where h and H are the hardening moduli, r_s (T) and R_s (T) are scalar functions describing the diffusion controlled 'static' or 'thermal' recovery, and r_d (T) and R_d (T) are the functions describing dynamic recovery.

If we assume that $W^p = 0$, we recover the Jaumann stress rate which results in the prediction of an oscillatory shear stress response in simple shear when coupled with a Prager kinematic hardening assumption [Johnson and Bammann 1984]. Alternatively we can choose,

$$W^p = R^T \dot{U} U^{-1} R \ ,$$

which recovers the Green-Naghdi rate of Cauchy stress and has been shown to be equivalent to Mandel's isoclinic state [Bammann and Aifantis 1987]. The model employing this rate allows a reasonable prediction of directional softening for some materials, but in general under-predicts the softening and does not accurately predict the axial stresses which occur in the torsion of the thin walled tube.

The final equation necessary to complete our description of high strain rate deformation is one which allows us to compute the temperature change during the deformation. In the absence of a coupled thermo-mechanical finite element code we assume adiabatic temperature change and follow the empirical assumption that 90 -95% of the plastic work is dissipated as heat. Hence,

$$\dot{T} = \frac{.9}{\rho C_v}\left(\sigma \cdot D^p\right),$$

where ρ is the density of the material and C_v the specific heat.

In terms of the input parameters the functions defined above become:

V(T) = C1 exp(-C2/T)	h(T) = C9 exp(C10/T)
Y(T) = C3 exp(C4/T)	rs(T) = C11exp(-C12/T)
f(T) = C5 exp(-C6/T)	RD(T) = C13exp(-C14/T)
rd(T) = C7 exp(-C8/T)	H(T) = C15exp(C16/T)
	RS(T) = C17exp(-C18/T)

and the heat generation coefficient is

$$HC = \frac{\dot{9}}{\rho G_v} \ .$$

*MAT_BAMMAN_DAMAGE

This is Material Type 52. This is an extension of model 51 which includes the modeling of damage. See [Bamman, et.al., 1990].

Card Format

Card 1

Variable	MID	RO	E	PR	T	HC		
Type	I	F	F	F	F	F		

Card 2

Variable	C1	C2	C3	C4	C5	C6	C7	C8
Type	F	F	F	F	F	F	F	F

Card 3

Variable	C9	C10	C11	C12	C13	C14	C15	C16
Type	F	F	F	F	F	F	F	F

Card 4

Variable	C17	C18	A1	A2	A3	A4	A5	A6
Type	F	F	F	F	F	F	F	F

Card 5

Variable	N	D0						
Type	F	F						

VARIABLE	DESCRIPTION
MID	Material identification. A unique number has to be chosen.
RO	Mass density
E	Young's modulus (psi)
PR	Poisson's ratio
T	Initial temperature (^{o}R)
HC	Heat generation coefficient ($^{o}R/psi$)
C1	Psi
C2	^{o}R
C3	Psi
C4	^{o}R
C5	$1/s$
C6	^{o}R
C7	$1/psi$
C8	^{o}R
C9	Psi
C10	^{o}R
C11	$1/psi\text{-}s$
C12	^{o}R

VARIABLE	DESCRIPTION
C13	$^1/_{psi}$
C14	^oR
C15	psi
C16	^oR
C17	$^1/_{psi\text{-}s}$
C18	^oR
A1	α_1, initial value of internal state variable 1
A2	α_2, initial value of internal state variable 2
A3	α_3, initial value of internal state variable 3
A4	α_4, initial value of internal state variable 4
A5	α_5, initial value of internal state variable 5
A6	α_6, initial value of internal state variable 6
N	Exponent in damage evolution
D0	Initial damage (porosity)

The evolution of the damage parameter, ϕ, is defined by [Bammann, et al. 1990]

$$\dot{\phi} = \beta\left[\frac{1}{(1-\phi)^N} - (1-\phi)\right]^{\left|D^p\right|}$$

in which

$$\beta = \sinh\left[\frac{2(2N-1)p}{(2N-1)\bar{\sigma}}\right]$$

where p is the pressure and $\bar{\sigma}$ is the effective stress.

*MAT_CLOSED_CELL_FOAM

This is Material Type 53. This allows the modeling of low density, closed cell polyurethane foam. It is for simulating impact limitors in automotive applications. The effect of the confined air pressure is included with the air being treated as an ideal gas. The general behavior is isotropic with uncoupled components of the stress tensor.

Card Format

Card 1

Variable	MID	RO	E	A	B	C	P0	PHI
Type	I	F	F	F	F	F	F	F

Card 2

Variable	GAMA0							
Type	F							

VARIABLE	DESCRIPTION
MID	Material identification. A unique number has to be chosen.
RO	Mass density
E	Young's modulus
A	a, factor for yield stress definition, see notes below.
B	b, factor for yield stress definition, see notes below.
C	c, factor for yield stress definition, see notes below.
P0	Initial foam pressure, P_0
PHI	Ratio of foam to polymer density, ϕ
GAMA0	Initial volumetric strain, γ_0. The default is zero.

A rigid, low density, closed cell, polyurethane foam model developed at Sandia Laboratories [Neilsen et al. 1987] has been recently implemented for modeling impact limiters in automotive applications. A number of such foams were tested at Sandia and reasonable fits to the experimental data were obtained.

In some respects this model is similar to the crushable honeycomb model type 26 in that the components of the stress tensor are uncoupled until full volumetric compaction is achieved. However, unlike the honeycomb model this material possesses no directionality but includes the effects of confined air pressure in its overall response characteristics..

$$\sigma_{ij} = \sigma_{ij}^{sk} - \delta_{ij}\sigma^{air}$$

where σ_{ij}^{sk} is the skeletal stress and σ^{air} is the air pressure computed from the equation:

$$\sigma^{air} = -\frac{p_0\gamma}{1+\gamma-\phi}$$

where p_0 is the initial foam pressure, usually taken as the atmospheric pressure, and γ defines the volumetric strain

$$\gamma = V - 1 + \gamma_0$$

where V is the relative volume, defined as the ratio of the current volume to the initial volume, and γ_0 is the initial volumetric strain, which is typically zero. The yield condition is applied to the principal skeletal stresses, which are updated independently of the air pressure. We first obtain the skeletal stresses:

$$\sigma_{ij}^{sk} = \sigma_{ij} + \sigma_{ij}\sigma^{air}$$

and compute the trial stress, σ^{skt}

$$\sigma_{ij}^{skt} = \sigma_{ij}^{sk} + E\dot{\varepsilon}_{ij}\,\Delta t$$

where E is Young's modulus. Since Poisson's ratio is zero, the update of each stress component is uncoupled and 2G=E where G is the shear modulus. The yield condition is applied to the principal skeletal stresses such that, if the magnitude of a principal trial stress component, σ_i^{skt}, exceeds the yield stress, σ_y, then

$$\sigma_i^{sk} = \min\left(\sigma_y, \left|\sigma_i^{skt}\right|\right)\frac{\sigma_i^{skt}}{\left|\sigma_i^{skt}\right|}$$

The yield stress is defined by

$$\sigma_y = a + b\,(1 + c\gamma)$$

where a, b, and c are user defined input constants and γ is the volumetric strain as defined above. After scaling the principal stresses they are transformed back into the global system.and the final stress state is computed

$$\sigma_{ij} = \sigma_{ij}^{sk} - \delta_{ij}\sigma^{air} \quad .$$

*MAT_ENHANCED_COMPOSITE_DAMAGE

These are Material Types 54-55 which are enhanced versions of the composite model material type 22. Arbitrary orthothropic materials, e.g., unidirectional layers in composite shell structures can be defined. Optionally, various types of failure can be specified following either the suggestions of [Chang and Chang, 1984] or [Tsai and Wu, 1981]. In addition special measures are taken for failure under compression. See [Matzenmiller and Schweizerhof, 1990]. This model is only valid for thin shell elements.

Card Format

Card 1

Variable	MID	RO	EA	EB	EC	PRBA	PRCA	PRCB
Type	I	F	F	F	F	F	F	F

Card 2

Variable	GAB	GBC	GCA	KF	AOPT			
Type	F	F	F	F	F			

Card 3

Variable	XP	YP	ZP	A1	A2	A3		
Type	F	F	F	F	F	F		

Card 4

Variable	V1	V2	V3	D1	D2	D3		
Type	F	F	F	F	F	F		

Card 5

Variable	TFAIL	XC	SOFT	FBRT				
Type	F	F	F	F				

Card 6

Variable	SC	XT	YT	YC	ALPH	CRIT		
Type	F	F	F	F	F	F		

VARIABLE	DESCRIPTION
MID	Material identification. A unique number has to be chosen.
RO	Mass density
EA	E_a, Young's modulus - longitudinal direction
EB	E_b, Young's modulus - transverse direction
EC	E_c, Young's modulus - normal direction
PRBA	ν_{ba}, Poisson's ratio ba
PRCA	ν_{ca}, Poisson's ratio ca
PRCB	ν_{cb}, Poisson's ratio cb
GAB	G_{ab}, shear modulus ab
GBC	G_{bc}, shear modulus bc
GCA	G_{ca}, shear modulus ca
KF	Bulk modulus of failed material

VARIABLE	DESCRIPTION
AOPT	Material axes option, see Figure 19.1: EQ. 0.0: locally orthotropic with material axes determined by element nodes as shown in Figure 19.1. Nodes 1, 2, and 4 of an element are identical to the Nodes used for the definition of a coordinate system as by *DEFINE_COORDINATE_NODES. EQ. 1.0: locally orthotropic with material axes determined by a point in space and the global location of the element center, this is the a-direction. EQ. 2.0: globally orthotropic with material axes determined by vectors defined below, as with *DEFINE_COORDINATE_VECTOR. EQ. 3.0: applicable to shell elements only. This option determines locally orthotropic material axes by offsetting the material axes by an angle to be specified from a line in the plane of the shell determined by taking the cross product of the vector **v** defined below with the shell normal vector.
XP YP ZP	Define coordinates of point p for AOPT = 1.
A1 A2 A3	Define components of vector a for AOPT = 2.
V1 V2 V3	Define components of vector v for AOPT = 3.
D1 D2 D3	Define components of vector d for AOPT = 2.
TFAIL	Time step size criteria for element deletion: EQ.≤ 0: no element deletion by time step size, EQ.0 ≤ t_{fail} ≤.1: element is deleted, when its time step is smaller than the given value, EQ.>.1: element is deleted, when the quotient of the actual time step and the original time step drops below the given value.
XC	Longitudinal compressive strength
SOFT	Softening reduction factor for material strength in crashfront elements (default = 1.0).
FBRT	Softening for fiber tensile strength: EQ.0.0: fiber rupture with tension cutoff GT:0.0: stress = FBRT * X_C after failure.
SC	Shear strength, ab plane, see below.
XT	Longitudinal tensile strength, see below.
YT	Transverse tensile strength, b-axis, see below.
YC	Transverse compressive strength, b-axis, see below 22.

VARIABLE	DESCRIPTION
ALPH	Shear stress parameter for the nonlinear term, see Material 22.
CRIT	Failure criterion (material number): EQ.54.0: Chang matrix failure criterion (as Material 22) (default), EQ.55.0: Tsai-Wu criterion for matrix failure.

The Chang/Chang criteria is given as follows:

for the tensile fiber mode,

$$\sigma_{aa} > 0 \quad then \quad e_f^2 = \left(\frac{\sigma_{aa}}{X_t}\right)^2 + \beta\left(\frac{\sigma_{ab}}{S_c}\right) - 1 \begin{cases} \geq 0 \; failed \\ < 0 \; elastic \end{cases},$$

$$E_a = E_b = G_{ab} = \nu_{ba} = \nu_{ab} = 0 \;,$$

for the compressive fiber mode,

$$\sigma_{aa} < 0 \quad then \quad e_c^2 = \left(\frac{\sigma_{aa}}{X_c}\right)^2 - 1 \begin{cases} \geq 0 \; failed \\ < 0 \; elastic \end{cases},$$

$$E_a = \nu_{ba} = \nu_{ab} = 0 .$$

for the tensile matrix mode,

$$\sigma_{bb} > 0 \quad then \quad e_m^2 = \left(\frac{\sigma_{bb}}{Y_t}\right)^2 + \left(\frac{\sigma_{ab}}{S_c}\right)^2 - 1 \begin{cases} \geq 0 \; failed \\ < 0 \; elastic \end{cases},$$

$$E_b = \nu_{ba} = 0. \quad \rightarrow G_{ab} = 0 \,,$$

and for the compressive matrix mode,

$$\sigma_{bb} < 0 \quad then \quad e_d^2 = \left(\frac{\sigma_{bb}}{2S_c}\right)^2 + \left[\left(\frac{Y_c}{2S_c}\right)^2 - 1\right]\frac{\sigma_{bb}}{Y_c} + \left(\frac{\sigma_{ab}}{S_c}\right)^2 - 1 \begin{cases} \geq 0 \; failed \\ < 0 \; elastic \end{cases},$$

$$_b = \nu_{ba} = \nu_{ab} = 0. \quad \rightarrow G_{ab} = 0$$
$$X_c = 2Y_c \quad for \; 50\% \; fiber \; volume .$$

In the Tsay/Wu criteria the tensile and compressive fiber modes are treated as in Chang and Chang. The failure criterion for the tensile and compressive matrix mode is given as:

$$e_{md}^2 = \frac{\sigma_{bb}^2}{Y_c Y_t} + \left(\frac{\sigma_{ab}}{S_c}\right)^2 + \frac{(Y_c - Y_t)\sigma_{bb}}{Y_c Y_t} - 1 \begin{cases} \geq 0 \; failed \\ < 0 \; elastic \end{cases}$$

For β =1 we get the original criterion of Hashin [1980]. For β =0 we get the maximum stress criterion which is found to compare better to experiments.

Tensile failure for each lamina can be brittle as indicated above if FBRT is set to 0.0. However, improved results are reported if the failed lamina carries the failure load or at least some part of it, until the entire laminate cross-section fails. Then FBRT is set to be larger than zero. With the value SOFT a degradation in strength is assumed for compression failure. If SOFT is set to be smaller than one, then the strength of the elements in the near neighborhood to the failed elements are multipled by the value of SOFT. This crudely accounts for the damage that occurs prior to failure and the tracking of the "crashfront."

*MAT_LOW_DENSITY_FOAM

This is Material Type 57. It is mainly for Modeling Low Density Urethane Foam, which is highly compressible. Its main applications are for seat cushions and padding on the Side Impact Dummies (SID). Optionally, a tension cut-off failure can be defined. Also, see the notes below.

Card Format

Card 1

Variable	MID	RO	E	LCID	TC	HU	BETA	DAMP
Type	I	F	F	F	F	F	F	F
Default	---	---	---	---	1.E+20	1.		
Remarks	---	---	---	---	---	3	1	---

Card 2

Variable	SHAPE	FAIL	BVFLAG	ED	BETA1	KCON		
Type	F	F	F	F	F	F		
Default	1.0	0.0	0.0	0.0	0.0	0.0		
Remarks	3	---	2					

VARIABLE	DESCRIPTION
MID	Material identification. A unique number has to be chosen.
RO	Mass density
E	Young's modulus

VARIABLE	DESCRIPTION
LCID	Load curve ID, see *DEFINE_CURVE, for nominal stress versus strain.
TC	Tension cut-off stress
HU	Hysteretic unloading factor between 0 and 1 (default=1, i.e., no energy dissipation), see also Figure 19.12.
BETA	β, decay constant to model creep in unloading
DAMP	Viscous coefficient (.05< recommended value <.50) to model damping effects.
SHAPE	Shape factor for unloading. Active for nonzero values of the hysteretic unloading factor. Values less than one reduces the energy dissipation and greater than one increases dissipation, see also Figure 19.12.
FAIL	Failure option after cutoff stress is reached: EQ.0.0: tensile stress remains at cut-off value, EQ.1.0: tensile stress is reset to zero.
BVFLAG	Bulk viscosity activation flag, see remark below: EQ.0.0: no bulk viscosity (recommended), EQ.1.0: bulk viscosity active.
ED	Optional Young's relaxation modulus, E_d, for rate effects. See comments below.
BETA1	Optional decay constant, β_1.
KCON	Stiffness coefficient for contact interface stiffness. Maximum slope in stress vs. strain curve is used. When the maximum slope is taken for the contact, the time step size for this material is reduced for stability. In some cases Δt may be significantly smaller, and defining a reasonable stiffness is recommended.

The compressive behavior is illustrated in Figure 19.12 where hysteresis on unloading is shown. This behavior under uniaxial loading is assumed not to significantly couple in the transverse directions. In tension the material behaves in a linear fashion until tearing occurs. Although our implementation may be somewhat unusual, it was motivated by Storakers [1986].

The model uses tabulated input data for the loading curve where the nominal stresses are defined as a function of the elongations, ε_i, which are defined in terms of the principal stretches, λ_i, as:

$$\varepsilon_i = \lambda_i - 1$$

The stretch ratios are found by solving for the eigenvalues of the left stretch tensor, V_{ij}, which is obtained via a polar decomposition of the deformation gradient matrix, F_{ij}. Recall that,

$$F_{ij} = R_{ik}U_{kj} = V_{ik}R_{kj}$$

The update of V_{ij} follows the numerically stable approach of [Taylor and Flanagan 1989]. After solving for the principal stretches, we compute the elongations and, if the elongations are compressive, the corresponding values of the nominal stresses, τ_i, are interpolated. If the elongations are tensile, the nominal stresses are given by

$$\tau_i = E\varepsilon_i$$

and the Cauchy stresses in the principal system become

$$\sigma_i = \frac{\tau_i}{\lambda_j \lambda_k}$$

The stresses can now be transformed back into the global system for the nodal force calculations.

Remarks:

1. When hysteretic unloading is used the reloading will follow the unloading curve if the decay constant, β, is set to zero. If β is nonzero the decay to the original loading curve is governed by the expression:

$$1.-e^{-\beta t}$$

2. The bulk viscosity, which generates a rate dependent pressure, may cause an unexpected volumetric response and, consequently, it is optional with this model.

3. The hysteretic unloading factor results in the unloading curve to lie beneath the loading curve as shown in Figure 19.12. This unloading provide energy dissipation which is reasonable in certains kinds of foam.

Rate effects are accounted for through linear viscoelasticity by a convolution integral of the form

$$\sigma_{ij}^r = \int_0^t g_{ijkl}(t-\tau)\frac{\partial \varepsilon_{kl}}{\partial \tau} d\tau$$

where $g_{ijkl}(t-\tau)$ is the relaxation function. The stress tensor, σ_{ij}^{r}, augments the stresses determined from the foam, σ_{ij}^{f}; consequently, the final stress, σ_{ij}, is taken as the summation of the two contributions:

$$\sigma_{ij} = \sigma_{ij}^{f} + \sigma_{ij}^{r}.$$

Since we wish to include only simple rate effects, the relaxation function is represented by one term from the Prony series:

$$g(t) = \alpha_0 + \sum_{m=1}^{N} \alpha_m e^{-\beta t}$$

given by,

$$g(t) = E_d e^{-\beta_1 t}$$

This model is effectively a Maxwell fluid which consists of a damper and spring in series. We characterize this in the input by a Young's modulus, E_d, and decay constant, β_1. The formulation is performed in the local system of principal stretches where only the principal values of stress are computed and triaxial coupling is avoided. Consequently, the one-dimensional nature of this foam material is unaffected by this addition of rate effects. The addition of rate effects necessitates twelve additional history variables per integration point. The cost and memory overhead of this model comes primarily from the need to "remember" the local system of principal stretches.

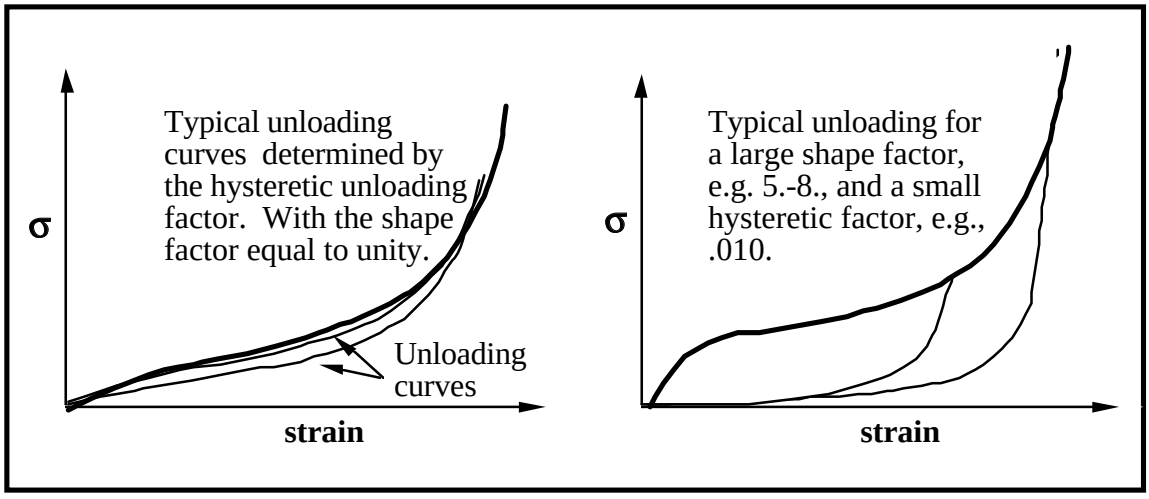

σ
Typical unloading curves determined by the hysteretic unloading factor. With the shape factor equal to unity.
Unloading curves
strain
σ
Typical unloading for a large shape factor, e.g. 5.-8., and a small hysteretic factor, e.g., .010.
strain

Figure 19.12. Behavior of the low density urethane foam model.

*MAT_COMPOSITE_FAILURE_MODEL

This is Material Type 59.

Card Format

Card 1

1	2	3	4	5	6	7	8
MID	RO	EA	EB	EC	PRBA	PRCA	PRCB
I	F	F	F	F	F	F	F

Card 2

GAB	GBC	GCA	KF	AOPT	MAFLAG		
F	F	F	F	F	F		

Card 3

XP	YP	ZP	A1	A2	A3		
F	F	F	F	F	F		

Card 4

V1	V2	V3	D1	D2	D3		
F	F	F	F	F	F		

Card 5

TSIZE	ALP	SOFT	FBRT	SR	SF		
F	F	F	F	F	F		

Card 6

Variable	XC	XT	YC	YT	SC			
Type	F	F	F	F	F			

VARIABLE	DESCRIPTION
MID	Material identification
RO	Density
EA	E_a, Young's modulus - longitudinal direction
EB	E_b, Young's modulus - transverse direction
EC	E_c, Young's modulus - normal direction
PRBA	ν_{ba}
PRCA	ν_{ca}
PRCB	ν_{cb}
GAB	G_{ab}
GBC	G_{bc}
GCA	G_{ca}
KF	Bulk modulus of failed material

VARIABLE	DESCRIPTION
AOPT	Material axes option: EQ.0.0: locally orthotropic with material axes determined by element nodes n_1, n_2, and n_4 as shown in Figure 19.1. EQ.1.0: locally orthotropic with material axes determined by a point in space and the global location of the element center. EQ.2.0: globally orthotropic with material axes determined by vectors defined below. EQ.3.0: applicable to shell elements only. This option determines locally orthotropic material axes by offsetting the material axes by an angle to be specified from a line in the plane of the shell determined by taking the cross product of the vector defined below with the shell normal vector.
MAFLAG	Material axes change flag for brick elements.
XP YP ZP	Define coordinates of point p for AOPT = 1.
A1 A2 A3	Define components of vector a for AOPT = 2.
V1 V2 V3	Define components of vector v for AOPT = 3.
D1 D2 D3	Define components of vector d for AOPT = 2: EQ.1.0: default, EQ.2.0: switch material axes a and b, EQ.3.0: switch material axes a and c.
TSIZE	Time step for automatic element deletion
ALP	Nonlinear shear stress parameter
SOFT	Softening reduction factor for strength in crush
FBRT	Softening of fiber tensile strength
SR	s_r, reduction factor(default=0.447)
SF	s_f, softening factor(default=0.0)
XC	Longitudinal compressive strength, a-axis
XT	Longitudinal tensile strength, a-axis
YC	Transverse compressive strength, b-axis
YT	Transverse tensile strength, b-axis
SC	Shear strength, ab plane: GT:0.0: faceted failure surface theory, LT:0.0: ellipsoidal failure surface theory.

*MAT_ELASTIC_WITH_VISCOSITY

This is Material Type 60 which was developed to simulate forming of glass products (e.g., car windshields) at high temperatures. Deformation is by viscous flow but elastic deformations can also be large. The material model, in which the viscosity may vary with temperature, is suitable for treating a wide range of viscous flow problems and is implemented for brick and shell elements.

Card Format

Card 1

Variable	MID	RO	V0	A	B	C	LCID	
Type	I	F	F	F	F	F	F	

Card 2

Variable	PR1	PR2	PR3	PR4	PR5	PR6	PR7	PR8
Type	F	F	F	F	F	F	F	F

Card 3

Variable	T1	T2	T3	T4	T5	T6	T7	T8
Type	F	F	F	F	F	F	F	F

Card 4

Variable	V1	V2	V3	V4	V5	V6	V7	V8
Type	F	F	F	F	F	F	F	F

Card 5

Variable	E1	E2	E3	E4	E5	E6	E7	E8
Type	F	F	F	F	F	F	F	F

Card 6

Variable	ALPHA1	ALPHA2	ALPHA3	ALPHA4	ALPHA5	ALPHA6	ALPHA7	ALPHA8
Type	F	F	F	F	F	F	F	F

VARIABLE	DESCRIPTION
MID	Material identification. A unique number has to be chosen.
RO	Mass density
V0	Temperature independent viscosity coefficient, V_0. If defined, the temperature dependent viscosity defined below is skipped, see type (i) and (ii) definitions for viscosity below.
A	Viscosity coefficient, see type (i) and (ii) definitions for viscosity below.
B	Viscosity coefficient, see type (i) and (ii) definitions for viscosity below.
C	Viscosity coefficient, see type (i) and (ii) definitions for viscosity below.
LCID	Load curve , see *DEFINE_CURVE, defining factor on viscosity versus time. (Optional).
T1, T2,...TN	Temperatures, define up to 8 values
PR1, PR2,...PRN	Poisson's ratios for the temperatures T_i
V1, V2,...VN	Corresponding viscosity coefficients (define only one if not varying with temperature).
E1, E2,...EN	Corresponding Young's moduli coefficients (define only one if not varying with temperature).
ALPHA....	Corresponding thermal expansion coefficients

This material model was developed to simulate forming of glass products (e.g. car windshields) at high temperatures. Deformation is by viscous flow but elastic deformations can also be large. The material model, in which the viscosity may vary with temperature, is suitable for treating a wide range of viscous flow problems and is implemented for brick and shell elements.

Volumetric behavior is treated as linear elastic. The deviatoric strain rate is considered to be the sum of elastic and viscous strain rates:

$$\underset{\sim}{\dot{\varepsilon}}'_{total} = \underset{\sim}{\dot{\varepsilon}}'_{elastic} + \underset{\sim}{\dot{\varepsilon}}'_{viscous} = \frac{\underset{\sim}{\sigma}'}{2G} + \frac{\underset{\sim}{\sigma}'}{2v}$$

where G is the elastic shear modulus, v is the viscosity coefficient, and ~ indicates a tensor. The stress increment over one timestep dt is

$$d\underset{\sim}{\sigma}' = 2G\,\underset{\sim}{\dot{\varepsilon}}'_{total}\,dt - \frac{G}{\upsilon}dt\,\underset{\sim}{\sigma}'$$

The stress before the update is used for $\underset{\sim}{\sigma}'$. For shell elements the through-thickness strain rate is calculated as follows.

$$d\sigma_{33} = 0 = K\left(\dot{\varepsilon}_{11} + \dot{\varepsilon}_{22} + \dot{\varepsilon}_{33}\right)dt + 2G\,\dot{\varepsilon}'_{33}\,dt - \frac{G}{\upsilon}dt\sigma'_{33}$$

where the subscript ij = 33 denotes the through-thickness direction and K is the elastic bulk modulus. This leads to:

$$\dot{\varepsilon}_{33} = -a\left(\dot{\varepsilon}_{11} + \dot{\varepsilon}_{22}\right) + bp$$

$$a = \frac{\left(K - \frac{2}{3}G\right)}{\left(K + \frac{4}{3}G\right)}$$

$$b = \frac{Gdt}{\upsilon\left(K + \frac{4}{3}G\right)}$$

in which p is the pressure defined as the negative of the hydrostatic stress.
The variation of viscosity with temperature can be defined in any one of the 3 ways.

(i) Constant, $V = V_0$ Do not define constants, A, B, and C or the piecewise curve.(leave card 4 blank)

(ii) $V = V_0 \infty 10^{**} (A/(T-B) + C)$

(iii) Piecewise curve: define the variation of viscosity with temperature.

Note: Viscosity is inactive during dynamic relaxation.

*MAT_KELVIN-MAXWELL_VISCOELASTIC

This is Material Type 61. It is a classical Kelvin-Maxwell model for modelling viscoelastic bodies, e.g., foams. Only valid for solid elements. See also notes below.

Card Format

Card 1

Variable	MID	RO	BULK	G0	GI	DC	FO	SO
Type	I	F	F	F	F	F	F	F
Default	none	none	none	none	none	0.0	0.0	0.0

VARIABLE	DESCRIPTION
MID	Material identification. A unique number has to be chosen.
RO	Mass density
BULK	Bulk modulus (elastic)
G0	Short-time shear modulus, G_0
GI	Long-time (infinite) shear modulus, G_∞
DC	Maxwell decay constant, β [FO=0.0] or Kelvin relaxation constant, τ [FO=1.0]
FO	Formulation option: EQ.0.0: Maxwell, EQ.1.0: Kelvin.
SO	Strain (logarithmic) output option to be plotted as component 7 in LS-TAURUS (D3PLOT file) which is the effective plastic strain component. The maximum values are updated for each element each time step: EQ.0.0: maximum principal strain that occurs during the calculation, EQ.1.0: maximum magnitude of the principal strain values that occurs during the calculation, EQ.2.0: maximum effective strain that occurs during the calculation.

The shear relaxation behavior is described for the Maxwell model by:

$$G(t) = G_\infty + (G_0 - G_\infty)\, e^{-\beta t}$$

A Jaumann rate formulation is used

$$\overset{\nabla}{\sigma}'_{ij} = 2\int_0^t G(t-\tau) D'_{ij}(\tau) dt$$

where the prime denotes the deviatoric part of the stress rate, $\overset{\nabla}{\sigma}_{ij}$, and the strain rate D_{ij}. For the Kelvin model the stress evolution equation is defined as:

$$\dot{s}_{ij} + \frac{1}{\tau} s_{ij} = \left(1 + \delta_{ij}\right) G_0 \dot{e}_{ij} + \left(1 + \delta_{ij}\right) \frac{G_\infty}{\tau} \dot{e}_{ij}$$

The strain data as given to LS-TAURUS may be used to predict damage, see [Bandak 1991].

*MAT_VISCOUS_FOAM

This is Material Type 62. This model was written to represent the energy absorbing foam found on certain crash dummies. This model was added to model the 'Confor Foam' on the ribs of the Eurosid. This model is only valid for solid elements mainly under compressive loading.

Card Format

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	E1	N1	V2	E2	N2	PR
Type	I	F	F	F	F	F	F	F

VARIABLE	DESCRIPTION
MID	Material identification. A unique number has to be chosen.
RO	Mass density
E1	Initial Young's modulus (E_1)
N1	Exponent in power law for Young's modulus (n_1)
V2	Viscous coefficient (V_2)
E2	Elastic modulus for viscosity (E_2), see notes below.
N2	Exponent in power law for viscosity (n_2)
PR	Poisson's ratio, ν

The model consists of a nonlinear elastic stiffness in parallel with a viscous damper. The elastic stiffness is intended to limit total crush while the viscosity absorbs energy. The stiffness E_2 exists to prevent timestep problems. It is used for time step calculations a long as E_1^t is smaller than E_2. It has to be carefully chosen to take into account the stiffening effects of the viscosity. Both E_1 and V_2 are nonlinear with crush as follows:

$$E_1^t = E_1\left(V^{-n_1}\right)$$

$$V_2^t = V_2\left(abs(1-V)\right)^{n_2}$$

where viscosity generates a shear stress given by

$$\tau = V_2\dot{\gamma}$$

$\dot{\gamma}$ is the engineering shear strain rate, and V is the relative volume defined by the ratio of the current to initial volume. Typical values are (units of N, mm, s)

E_1=0.0036	n_1=4.0	V_2 =0.0015
E_2=100.0	n_2=0.2	ν =0.05

*MAT_CRUSHABLE_FOAM

This is Material Type 63 which is dedicated to modeling crushable foam with optional damping and tension cutoff. Unloading is fully elastic. Tension is treated as completely elastic-plastic.

Card Format

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	E	PR	LCID	TSC	DAMP	
Type	I	F	F	F	F	F	F	

VARIABLE	DESCRIPTION
MID	Material identification. A unique number has to be chosen.
RO	Mass density
E	Young's modulus
PR	Poisson's ratio
LCID	Load curve ID defining yield stress versus volumetric strain, γ, see Figure 19.13.
TSC	Tensile stress cutoff
DAMP	Rate senitivity via damping coefficient (.05<recommended value<.50).

The volumetric strain is defined in terms of the relative volume, V, as:

$$\gamma = 1.-V$$

The relative volume is defined as the ratio of the current to the initial volume.

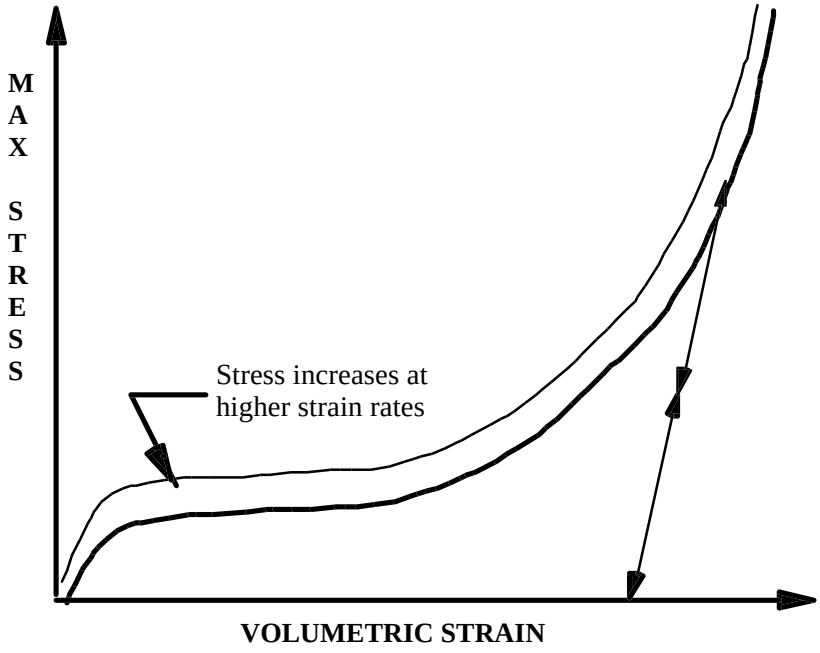


Figure 19.13. Behavior of strain rate sensitive crushable foam. Unloading is elastic to the tension cutoff. Subsequent reloading follows the unloading curve.

*MAT_RATE_SENSITIVE_POWERLAW_PLASTICITY

This is Material Type 64 which will model strain rate sensitive elasto-plastic material with a power law hardening. Optionally, the coefficients can be defined as functions of the effective plastic strain.

Card Format

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	E	PR	K	M	N	E0
Type	I	F	F	F	F	F	F	F
Default	---	---	---	---	---	0.0001	---	0.0002

VARIABLE	DESCRIPTION
MID	Material identification. A unique number has to be chosen.
RO	Mass density
E	Young's modulus of elasticity
PR	Poisson's ratio
K	Material constant, k. If k<0 the absolute value of k is taken as the load curve number that defines k as a function of effective plastic strain.
M	Strain hardening coefficient, m. If m<0 the absolute value of m is taken as the load curve number that defines m as a function of effective plastic strain.
N	Strain rate sensitivity coefficient, n. If n<0 the absolute value of n is taken as the load curve number that defines n as a function of effective plastic strain.
E0	Initial strain rate (default = 0.0002)

This material model follows a constitutive relationship of the form:

$$\sigma = k\varepsilon^m \dot{\varepsilon}^n$$

where the constants k, m, and n can be expressed as functions of strain or can be constant with respect to the plastic strain. The case of no strain hardening can be obtained by setting the exponent of the plastic strain equal to a very small positive value, i.e. 0.0001.

*MAT_MODIFIED_ZERILLI_ARMSTRONG

This is Material Type 65 which is a rate and temperature sensitive plasticity model which is sometimes preferred in ordnance design calculations.

Card Format

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	G	E0	N	TROOM	PC	SPALL
Type	I	F	F	F	F	F	F	F

Card 2	1	2	3	4	5	6	7	8
Variable	C1	C2	C3	C4	C5	C6	EFAIL	
Type	F	F	F	F	F	F	F	

Card 3	1	2	3	4	5	6	7	8
Variable	B1	B2	B3	G1	G2	G3	G4	
Type	F	F	F	F	F	F	F	

VARIABLE	DESCRIPTION
MID	Material identification. A unique number has to be chosen.
RO	Mass density
G	Shear Modulus
E0	$\varepsilon_{0,}$ initial strain
N	n, exponent for Bcc metal
TROOM	Room temperature, T_r

VARIABLE	DESCRIPTION
PC	Pressure cutoff, pc
SPALL	Spall Type: EQ.1.0: minimum pressure limit, EQ.2.0: maximum principal stress, EQ.3.0: minimum pressure cutoff.
C1	C_1, coefficients for flow stress, see notes below.
C2	C_2, coefficients for flow stress, see notes below.
C3	C_3, coefficients for flow stress, see notes below.
C4	C_4, coefficients for flow stress, see notes below.
C5	C_5, coefficients for flow stress, see notes below.
C6	C_6, coefficients for flow stress, see notes below.
EFAIL	Failure strain for erosion
B1	B_1, coefficients for polynomial to represent temperature dependency of flow stress yield.
B2	B_2
B3	B_3
G1	G_1, coefficients for defining heat capacity and temperature dependency of heat capacity.
G2	G_2
G3	G_3
G4	G_4

The Armstrong-Zerilli Material Model express the yield stress as follows.

For FCC metals,

$$\sigma = C_1 + \left(C_2 \varepsilon^{1/2} \left[e^{\left(-C_3 T + C_4 \ln(\dot{\varepsilon}) \right)} \right] + C_5 \right) \left(\frac{\mu(T)}{\mu(293)} \right)$$

and for BCC metals,

$$\sigma = C_1 + C_2 e^{\left(C_3 + C_4 \ln(\varepsilon)\right)} + \left(C_5 \varepsilon^n + C_6\right)\left(\frac{\mu(T)}{\mu(293)}\right)$$

where

$$\left(\frac{\mu(T)}{\mu(293)}\right) = B_1 + B_2 T + B_3 T^2$$

The relationship between heat capacity and temperature may be characterized by a cubic polynomial equation as follows:

$$C_p = G_1 + G_2 T + G_3 T^2 + G_4 T^3$$

*MAT_LINEAR_ELASTIC_DISCRETE_BEAM

This is Material Type 66. This material model is defined for simulating the effects of a linear elastic zero length beams by using six springs each acting about one of the six local degrees-of-freedom. Translational/rotational stiffness and viscous damping effects are considered for a local cartesian system, see notes below. Applications for this element include the modeling of joint stiffnesses.

Card Format

Card 1

Variable	MID	RO	TKR	TKS	TKT	RKR	RKS	RKT
Type	I	F	F	F	F	F	F	F

Card 2

Variable	TDR	TDS	TDT	RDR	RDS	RDT		
Type	F	F	F	F	F	F		

VARIABLE	DESCRIPTION
MID	Material identification. A unique number has to be chosen.
RO	Mass density, see also “volume” in the *SECTION_BEAM definition.
TKR	Translational stiffness about local r-axis, see notes below.
TKS	Translational stiffness about local s-axis
TKT	Translational stiffness about local t-axis
RKR	Rotational stiffness about the local r-axis
RKS	Rotational stiffness about the local s-axis
RKT	Rotational stiffness about the local t-axis
TDR	Translational viscous damper about local r-axis. (Optional).
TDS	Translational viscous damper about local s-axis. (Optional).

VARIABLE	DESCRIPTION
TDT	Translational viscous damper about local t-axis. (Opitonal).
RDR	Rotational viscous damper about the local r-axis. (Optional).
RDS	Rotational viscous damper about the local s-axis. (Optional).
RDT	Rotational viscous damper about the local t-axis. (Optional).

The formulation of the discrete beam (type 6) assumes that the beam is of zero length and requires no orientation node. A small distance between the nodes joined by the beam is permitted. The local coordinate system which determines (r,s,t) is given by the coordinate ID, see *DEFINE_COORDINATE_OPTION, in the cross sectional input, see *SECTION_BEAM, where the global system is the default.

For null stiffness coefficients, no forces corresponding to these null values will develop. The viscous damping coefficients are optional.

*MAT_NONLINEAR_ELASTIC_DISCRETE_BEAM

This is Material Type 67. This material model is defined for simulating the effects of nonlinear elastic and nonlinear viscous zero length beams by using six springs each acting about one of the six local degrees-of-freedom. Arbitrary curves to model transitional/ rotational stiffness and damping effects are allowed. See notes below.

Card Format

Card 1

Variable	MID	RO	LCIDTR	LCIDTS	LCIDTT	LCIDRR	LCIDRS	LCIDRT
Type	I	F	F	F	F	F	F	F

Card 2

Variable	LCIDTDR	LCIDTDS	LCIDTDT	LCIDRDR	LCIDRDS	LCIDRDT		
Type	F	F	F	F	F	F		

VARIABLE	DESCRIPTION
MID	Material identification. A unique number has to be chosen.
RO	Mass density, see also volume in *SECTION_BEAM definition.
LCIDTR	Load curve ID defining translational force resultant along local r-axis versus relative translational displacement, see Figure 19.14.
LCIDTS	Load curve ID defining translational force resultant along local s-axis versus relative translational displacement.
LCIDTT	Load curve ID defining translational force resultant along local t-axis versus relative translational displacement.
LCIDRR	Load curve ID defining rotational moment resultant about local r-axis versus relative rotational displacement.
LCIDRS	Load curve ID defining rotational moment resultant about local s-axis versus relative rotational displacement.

VARIABLE	DESCRIPTION
LCIDRT	Load curve ID defining rotational moment resultant about local t-axis versus relative rotational displacement.
LCIDTDR	Load curve ID defining translational damping force resultant along local r-axis versus relative translational velocity.
LCIDTDS	Load curve ID defining translational damping force resultant along local s-axis versus relative translational velocity.
LCIDTDT	Load curve ID defining translational damping force resultant along local t-axis versus relative translational velocity.
LCIDRDR	Load curve ID defining rotational damping moment resultant about local r-axis versus relative rotational velocity.
LCIDRDS	Load curve ID defining rotational damping moment resultant about local s-axis versus relative rotational velocity.
LCIDRDT	Load curve ID defining rotational damping moment resultant about local t-axis versus relative rotational velocity.

For null load curve ID's, no forces are computed.

The formulation of the discrete beam (type 6) assumes that the beam is of zero length and requires no orientation node. A small distance between the nodes joined by the beam is permitted. The local coordinate system which determines (r,s,t) is given by the coordinate ID, see *DEFINE_ COORDINATE_OPTION, in the cross sectional input, see *SECTION_BEAM, where the global system is the default.

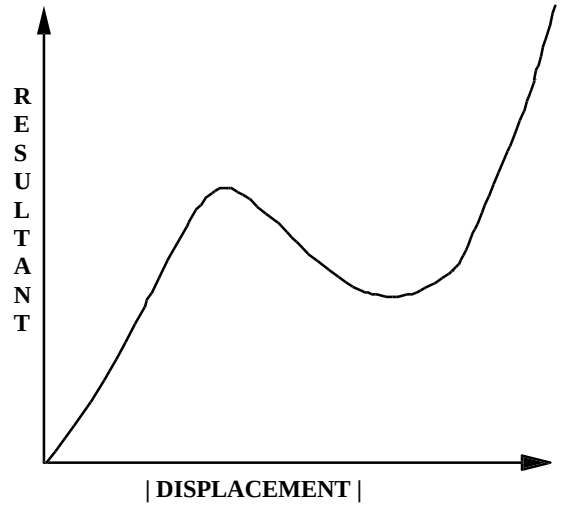


Figure 19.14. The resultant forces and moments are determined by a table lookup. The origin of the load curve is at [0,0] and tension and compression are similarly treated.

*MAT_NONLINEAR_PLASTIC_DISCRETE_BEAM

This is Material Type 68. This material model is defined for simulating the effects of nonlinear elastoplastic, linear viscous behavior of zero length beams by using six springs each acting about one of the six local degrees-of-freedom. Translational/rotational stiffness and damping effects can be considered. The plastic behavior is modelled using force/moment curves versus displacements/ rotation. Optionally, failure can be specified based on a force/moment criterion and a displacement/ rotation criterion. See also notes below.

Card Format

Card 1

Variable	MID	RO	TKR	TKS	TKT	RKR	RKS	RKT
Type	I	F	F	F	F	F	F	F
Default	none	none	none	none	none	none	none	none

Card 2

Variable	TDR	TDS	TDT	RDR	RDS	RDT		
Type	F	F	F	F	F	F		
Default	none	none	none	none	none	none		

Card 3

Variable	LCPDR	LCPDS	LCPDT	LCPMR	LCPMS	LCPMT		
Type	F	F	F	F	F	F		
Default	0.0	0.0	0.0	0.0	0.0	0.0		

Card 4

Variable	FFAILR	FFAILS	FFAILT	MFAILR	MFAILS	MFAILT		
Type	F	F	F	F	F	F		
Default	0.0	0.0	0.0	0.0	0.0	0.0		

Card 5

Variable	UFAILR	UFAILS	UFAILT	TFAILR	TFAILS	TFAILT		
Type	F	F	F	F	F	F		
Default	0.0	0.0	0.0	0.0	0.0	0.0		

VARIABLE	DESCRIPTION
MID	Material identification. A unique number has to be chosen.
RO	Mass density, see also volume on *SECTION_BEAM definition.
TKR	Translational stiffness about local r-axis
TKS	Translational stiffness about local s-axis
TKT	Translational stiffness about local t-axis
RKR	Rotational stiffness about the local r-axis
RKS	Rotational stiffness about the local s-axis
RKT	Rotational stiffness about the local t-axis
TDR	Translational viscous damper about local r-axis
TDS	Translational viscous damper about local s-axis
TDT	Translational viscous damper about local t-axis
RDR	Rotational viscous damper about the local r-axis

VARIABLE	DESCRIPTION
RDS	Rotational viscous damper about the local s-axis
RDT	Rotational viscous damper about the local t-axis
LCPDR	Load curve ID-yield force versus plastic displacement r-axis
LCPDS	Load curve ID-yield force versus plastic displacement s-axis
LCPDT	Load curve ID-yield force versus plastic displacement t-axis
LCPMR	Load curve ID-yield moment versus plastic rotation r-axis
LCPMS	Load curve ID-yield moment versus plastic rotation s-axis
LCPMT	Load curve ID-yield moment versus plastic rotation t-axis
FFAILR	Optional failure parameter. If zero, the corresponding force, F_r, is not considered in the failure calculation.
FFAILS	Optional failure parameter. If zero, the corresponding force, F_s, is not considered in the failure calculation.
FFAILT	Optional failure parameter. If zero, the corresponding force, F_t, is not considered in the failure calculation.
MFAILR	Optional failure parameter. If zero, the corresponding moment, M_r, is not considered in the failure calculation.
MFAILS	Optional failure parameter. If zero, the corresponding moment, M_s, is not considered in the failure calculation.
MFAILT	Optional failure parameter. If zero, the corresponding moment, M_t, is not considered in the failure calculation.
UFAILR	Optional failure parameter. If zero, the corresponding displacement, u_r, is not considered in the failure calculation.
UFAILS	Optional failure parameter. If zero, the corresponding displacement, u_s, is not considered in the failure calculation.
UFAILT	Optional failure parameter. If zero, the corresponding displacement, u_t, is not considered in the failure calculation.
TFAILR	Optional failure parameter. If zero, the corresponding rotation, θ_r, is not considered in the failure calculation.
TFAILS	Optional failure parameter. If zero, the corresponding rotation, θ_s, is not considered in the failure calculation.

VARIABLE	DESCRIPTION
TFAILT	Optional failure parameter. If zero, the corresponding rotation, θ_t, is not considered in the failure calculation.

For the translational and rotational degrees of freedom where elastic behavior is desired, set the load curve ID to zero.

The formulation of the discrete beam (type 6) assumes that the beam is of zero length and requires no orientation node. A small distance between the nodes joined by the beam is permitted. The local coordinate system which determines (r,s,t) is given by the coordinate ID (see *DEFINE_COORDINATE_OPTION) in the cross sectional input, see *SECTION_BEAM, where the global system is the default.

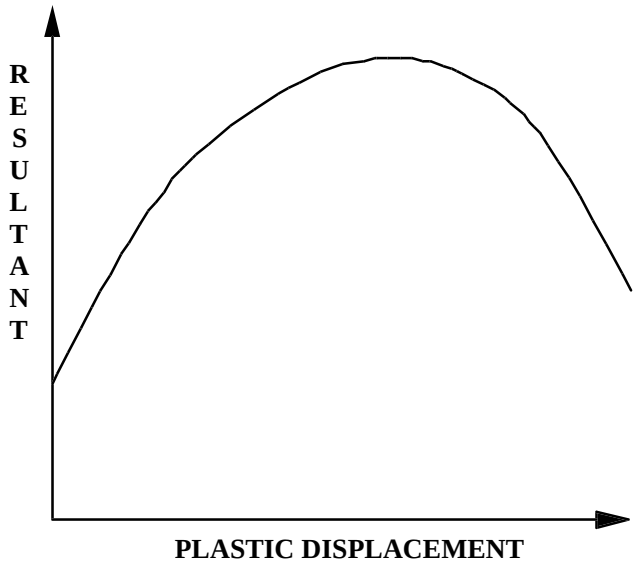


Figure 19.15. The resultant forces and moments are limited by the yield definition. The initial yield point corresponds to a plastic displacement of zero.

Catastrophic failure based on force resultants occurs if the following inequality is satisfied.

$$\left(\frac{F_r}{F_r^{fail}}\right)^2+\left(\frac{F_s}{F_s^{fail}}\right)^2+\left(\frac{F_t}{F_t^{fail}}\right)^2+\left(\frac{M_r}{M_r^{fail}}\right)^2+\left(\frac{M_s}{M_s^{fail}}\right)^2+\left(\frac{M_t}{M_t^{fail}}\right)^2-1.\geq 0.$$

After failure the discrete element is deleted. Likewise, catastrophic failure based on displacement resultants occurs if the following inequality is satisfied:

$$\left(\frac{u_r}{u_r^{fail}}\right)^2+\left(\frac{u_s}{u_s^{fail}}\right)^2+\left(\frac{u_t}{u_t^{fail}}\right)^2+\left(\frac{\theta_r}{\theta_r^{fail}}\right)^2+\left(\frac{\theta_s}{\theta_s^{fail}}\right)^2+\left(\frac{\theta_t}{\theta_t^{fail}}\right)^2-1.\geq 0.$$

After failure the discrete element is deleted. If failure is included either one or both of the criteria may be used.

*MAT_SID_DAMPER_DISCRETE_BEAM

This is Material Type 69. The side impact dummy uses a damper that is not adequately treated by the nonlinear force versus relative velocity curves since the force characteristics are dependent on the displacement of the piston. See also notes below.

Card Format

Card 1

Variable	MID	RO	ST	D	R	H	K	C
Type	I	F	F	F	F	F	F	F

Card 2

Variable	C3	STF	RHOF	C1	C2	LCIDF	LCIDD	S0
Type	F	F	F	F	F	F	F	F

Read in up to 16 orifice locations with orifice location per card. Input is terminated when a "*" card is found.

Cards 3,...

Variable	ORFLOC	ORFRAD						
Type	F	F						

VARIABLE	DESCRIPTION
MID	Material identification. A unique number has to be chosen.
RO	Mass density, see also volume on *SECTION_BEAM definition.
ST	S_t, piston stroke. S_t must equal or exceed the length of the beam element, see Figure 19.16 below.
D	d, piston diameter
R	R, default orifice radius
H	h, orifice controller position
K	K, damping constant
	LT.0.0: |K| is the load curve number ID, see *DEFINE_CURVE, defining the damping coefficient as a function of the absolute value of the relative velocity.
C	C, discharge coefficient
C3	Coefficient for fluid inertia term
STF	k, stiffness coefficient if piston bottoms out
RHOF	ρ_{fluid}, fluid density
C1	C_1, coefficient for linear velocity term
C2	C_2, coefficient for quadratic velocity term
LCIDF	Load curve number ID defining force versus piston displacement, s, i.e., term $f(s+s_0)$. Compressive behavior is defined in the positive quadrant of the force displacement curve. Displacements falling outside of the defined force displacement curve are extrapolated. Care must be taken to ensure that extrapolated values are reasonable.
LCIDD	Load curve number ID defining damping coefficient versus piston displacement, s, i.e., $g(s+s_0)$. Displacements falling outside the defined curve are extrapolated. Care must be taken to ensure that extrapolated values are reasonable.
S0	Initial displacement s_0, typically set to zero. A positive displacement corresponds to compressive behavior.
ORFLOC	d_i, orifice location of ith orifice relative to the fixed end.
ORFLOC	r_i, orifice radius of ith orifice, if zero the default radius is used.

As the damper moves, the fluid flows through the open orifices to provide the necessary damping resistance. While moving as shown in Figure 19.16, the piston gradually blocks off and effectively closes the orifices. The number of orifices and the size of their opening control the damper resistance and performance. The damping force is computed from,

$$F = KA_pV_p\left\{\frac{C_1}{A_0^t} + C_2|V_p|\rho_{fluid}\left[\left(\frac{A_p}{CA_0^t}\right)^2 - 1\right]\right\} - f(s+s_0) + V_p g(s+s_0)$$

where K is a user defined constant or a tabulated function of the absolute value of the relative velocity, V_p is the piston velocity, C is the discharge coefficient, A_p is the piston area, A_0^t is the total open areas of orifices at time t, ρ_{fluid} is the fluid density, C_1 is the coefficient for the linear term, and C_2 is the coefficient for the quadratic term.

In the implementation the orifices are assumed to be circular with partial covering by the orifice controller. The gradually shutdown as the piston closes of the orifice is properly taken into account. If the piston stroke is exceeded, the stiffness value defined by STF stops further movement. The piston stroke must exceed the initial length of the beam element. The time step calculation is also based on the stiffness value. A typical force versus displacement curve at constant relative velocity is shown in Figure 19.17.

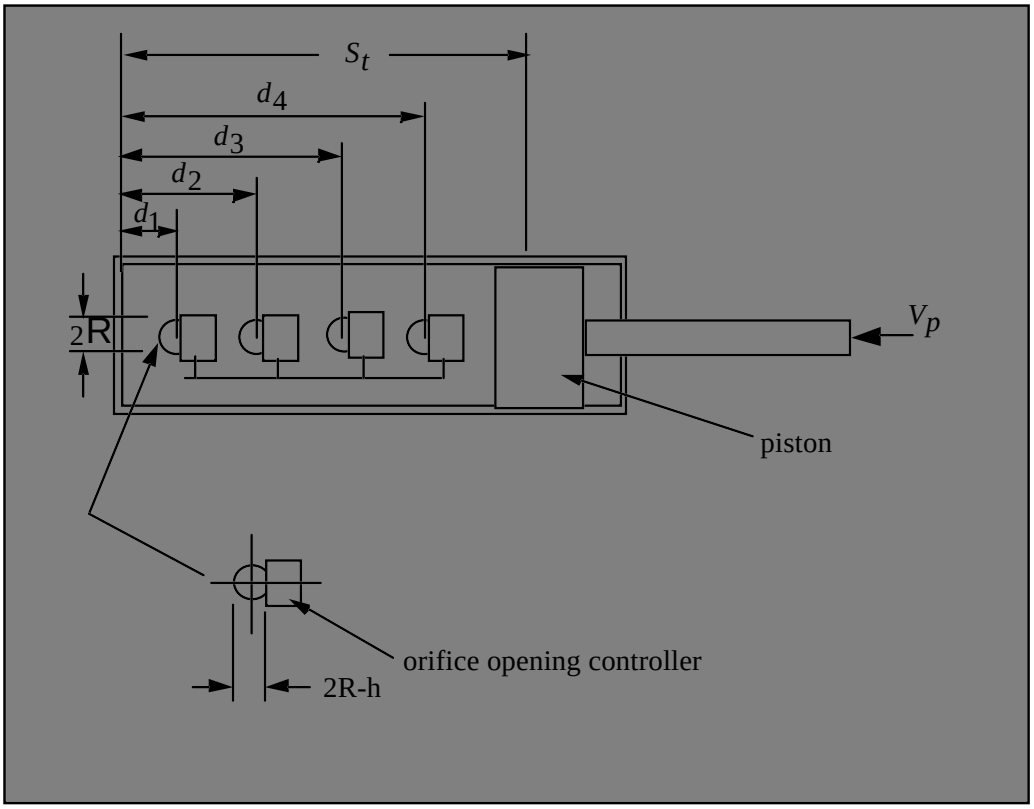


Figure 19.16. Mathematical model for the Side Impact Dummy damper.

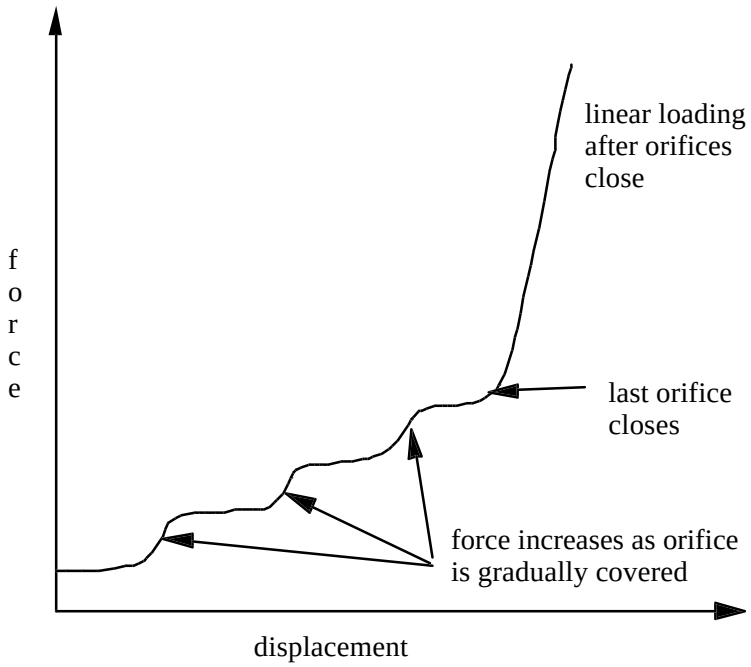


Figure 19.17. Force versus displacement as orifices are covered at a constant relative velocity. Only the linear velocity term is active.

***MAT_HYDRAULIC_GAS_DAMPER_DISCRETE_BEAM**

This is Material Type 70. This special purpose element represents a combined hydraulic and gas-filled damper which has a variable orifice coefficient. A schematic of the damper is shown in Figure 19.18. Dampers of this type are sometimes used on buffers at the end of railroad tracks and as aircraft undercarriage shock absorbers. This material can be used only as a discrete beam element. See also notes below.

Card Format

Card 1

Variable	MID	RO	CO	N	P0	PA	AP	KH
Type	I	F	F	F	F	F	F	F

Card 2

Variable	LCID	FR	SCLF	CLEAR				
Type	F	F	F	F				

VARIABLE	DESCRIPTION
MID	Material identification. A unique number has to be chosen.
RO	Mass density, see also volume in *SECTION_BEAM defintion.
CO	Length of gas column, C_o
N	Adiabatic constant
P0	Initial gas pressure, P_0
PA	Atmospheric pressure, P_a
AP	Piston cross sectional area, A_p
KH	Hydraulic constant, K

VARIABLE	DESCRIPTION
LCID	Load curve ID, see *DEFINE_CURVE, defining the orifice area, a_0, versus element deflection.
FR	Return factor on orifice force. This acts as a factor on the hydraulic force only and is applied when unloading. It is intended to represent a valve that opens when the piston unloads to relieve hydraulic pressure. Set it to 1.0 for no such relief.
SCLF	Scale factor on force. (Default = 1.0)
CLEAR	Clearance (if nonzero, no tensile force develops for positive displacements and negative forces develop only after the clearance is closed.

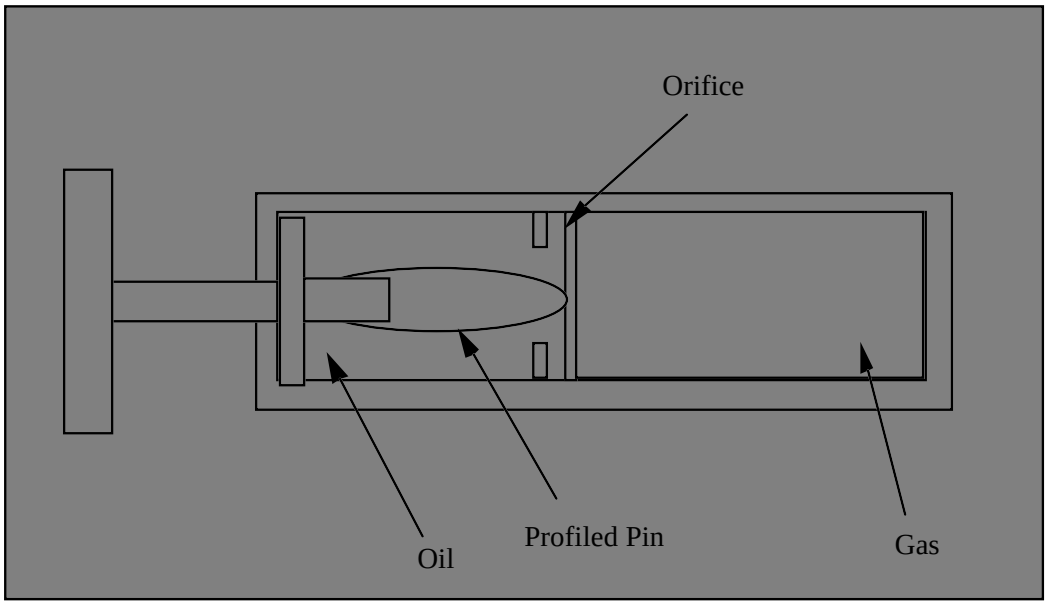


Figure 19.18. Schematic of Hydraulic/Gas damper.

As the damper compresses two actions contribute to the force developed. First, the gas is adiabatically compressed into a smaller volume. Secondly, oil is forced through an orifice. A profiled pin may occupy some of the cross-sectional area of the orifice; thus, the orifice area available for the oil varies with the stroke. The force is assumed proportional to the square of the velocity and inversely proportional to the available area.

The equation for this element is:

$$F = SCLF \cdot \left\{ K_h \left(\frac{V}{a_o} \right)^2 + \left[P_0 \left(\frac{C_0}{C_0 - S} \right)^n - P_a \right] \cdot A_p \right\}$$

where S is the element deflection and V is the relative velocity across the element.

*MAT_CABLE_DISCRETE_BEAM

This is Material Type 71. This model permits elastic cables to be realistically modelled; thus, no force will develop in compression.

Card Format

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	E	LCID				
Type	I	F	F	F				

VARIABLE	DESCRIPTION
MID	Material identification. A unique number has to be chosen.
RO	Mass density, see also volume in *SECTION_BEAM definition.
E	Young's modulus
LCID	Load curve ID, see *DEFINE_CURVE, defining the stress versus engineering strain. (Optional).

The force, F, generated by the cable is nonzero if and only if the cable is tension. The force is given by:

$$F = K \cdot \max(\Delta L, 0.)$$

where ΔL is the change in length

$$\Delta L = \textit{current length} - (\textit{initial length} - \textit{offset})$$

and the stiffness is defined as:

$$K = \frac{E \cdot \textit{area}}{(\textit{initial length} - \textit{offset})}$$

The area and offset are defined on either the cross section or element cards. For a slack cable the offset should be input as a negative length. For an initial tensile force the offset should be positive.

If a load curve is specified the Young's modulus will be ignored and the load curve will be used instead. The points on the load curve are defined as engineering stress versus engineering strain, i.e., the change in length over the initial length. The unloading behavior follows the loading.

*MAT_BILKHU/DUBOIS_FOAM

This is Material Type 75. This model is for the simulation of isotropic crushable forms. Uniaxial and triaxial test data have to be used. For the elastic response, the Poisson ratio is set to zero.

Card Format

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	YM	LCPY	LCUYS	VC		
Type	I	F	F	F	F	F		

VARIABLE	DESCRIPTION
MID	Material identification. A unique number has to be chosen.
RO	Mass density
YM	Young's modulus (E)
LCPY	Load curve ID giving pressure for plastic yielding versus volumetric strain, see Figure 19.19.
LCUYS	Load curve ID giving unixial yield stress versus volumetric strain, see Figure 19.19.
VC	Viscous damping coefficient (.05<recommended value<.50).

The logarithmic volumetric strain is defined in terms of the relative volume, V, as:

$$\gamma = -\ln(V)$$

In defining the curves the stress and strain pairs should be positive values starting with a volumetric strain value of zero.

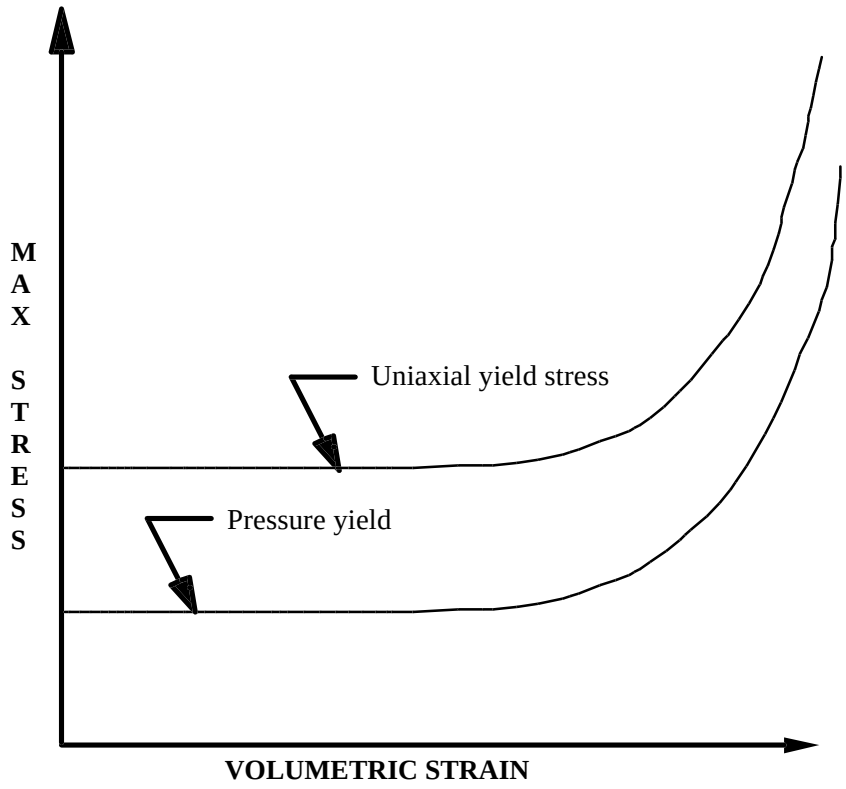


Figure 19.19. Behavior of crushable foam. Unloading is elastic.

The yield surface is defined as an ellipse in the equivalent pressure and von Mises stress plane.

*MAT_GENERAL_VISCOELASTIC

This is Material Type 76. This material model provides a general viscoelastic Maxell model having up to 6 terms in the prony series expansion and is useful for modeling dense continuum rubbers and solid explosives. Either the coefficients of the prony series expansion or a relaxation curve may be specified to define the viscoelastic deviatoric and bulk behavior.

Card Format

Card 1

Variable	MID	RO	BULK					
Type	I	F	F					

Insert a blank card here if constants are defined on cards 3,4,... below.

Card 2

Variable	LCID	NT	BSTART	TRAMP	LCIDK	NTK	BSTARTK	TRAMPK
Type	I	I	F	F	I	I	F	F

Card Format for viscoelastic constants. Up to 6 cards may be input. A keyword card (with a "*" in column 1) terminates this input if less than 6 cards are used. These cards are not needed if relaxation data is defined. The number of terms for the shear behavior may differ from that for the bulk behavior: simply insert zero if a term is not included.

Optional Cards

Variable	GI	BETAI	KI	BETAKI				
Type	F	F	F	F				

VARIABLE	DESCRIPTION
MID	Material identification. A unique number has to be chosen.
RO	Mass density.
BULK	Elastic bulk modulus.
LCID	Load curve ID for deviatoric behavior if constants, G_i, and β_i are determined via a least squares fit. This relaxation curve is shown below.
NT	Number of terms in shear fit. If zero the default is 6. Currently, the maximum number is set to 6.
BSTART.	In the fit, β_1 is set to zero, β_2 is set to BSTART, β_3 is 10 times β_2 , β_4 is 100 times greater than β_3 , and so on. If zero, BSTART= .01.
TRAMP	Optional ramp time for loading.
LCIDK	Load curve ID for bulk behavior if constants, K_i, and $\beta\kappa_i$ are determined via a least squares fit. This relaxation curve is shown below.
NTK	Number of terms desired in bulk fit. If zero the default is 6. Currently, the maximum number is set to 6.
BSTARTK.	In the fit, $\beta\kappa_1$ is set to zero, $\beta\kappa_2$ is set to BSTARTK, $\beta\kappa_3$ is 10 times $\beta\kappa_2$, $\beta\kappa_4$ is 100 times greater than $\beta\kappa_3$, and so on. If zero, BSTARTK= .01.
TRAMPK	Optional ramp time for bulk loading.
GI	Optional shear relaxation modulus for the ith term
BETAI	Optional shear decay constant for the ith term
KI	Optional bulk relaxation modulus for the ith term
BETAKI	Optional bulk decay constant for the ith term

Rate effects are taken into accounted through linear viscoelasticity by a convolution integral of the form:

$$\sigma_{ij} = \int_0^t g_{ijkl}(t-\tau)\frac{\partial \varepsilon_{kl}}{\partial \tau}\, d\tau$$

where $g_{ijkl}(t-\tau)$ is the relaxation functions for the different stress measures. This stress is addedto the stress tensor determined from the strain energy functional.

If we wish to include only simple rate effects, the relaxation function is represented by six terms from the Prony series:

$$g(t) = \sum_{m=1}^{N} G_m e^{-\beta_m t}$$

We characterize this in the input by shear modulii, G_i, and decay constants, β_i. An arbitrary number of terms, up to 6, may be used when applying the viscoelastic model.

For volumetric relaxation, the relaxation function is also represented by the Prony series in terms of bulk modulii:

$$k(t) = \sum_{m=1}^{N} K_m e^{-\beta_{k_m} t}$$

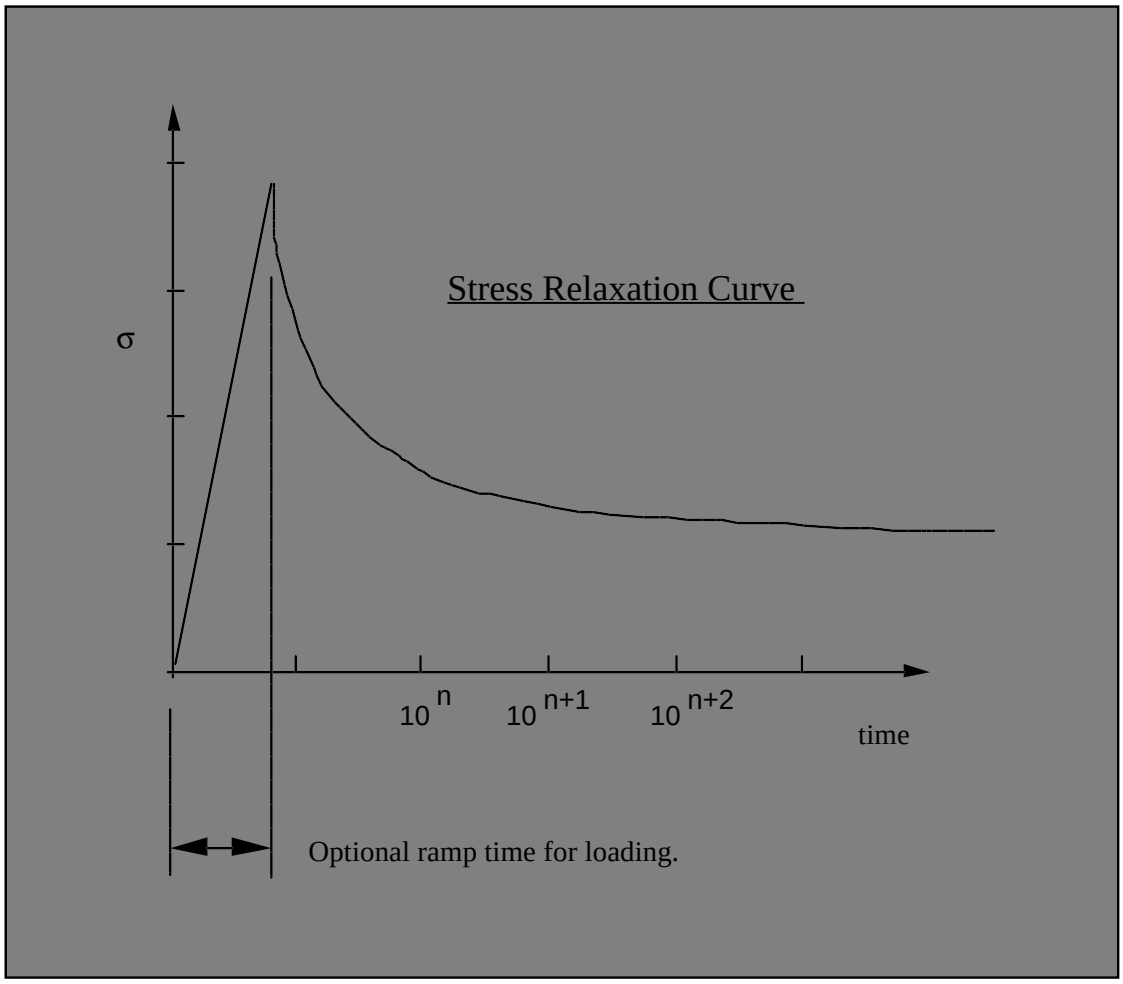


Figure 19.20. Relaxation curve. This curve defines stress versus time where time is defined on a logarithmic scale. For best results, the points defined in the load curve should be equally spaced on the logarithmic scale. Furthermore, the load curve should be smooth and defined in the positive quadrant. If nonphysical values are determined by least squares fit, LS-DYNA3D will terminate with an error message after the initialization phase is completed. If the ramp time for loading is included, then the relaxation which occurs during the loading phase is taken into account. This effect may or may not be important.

*MAT_HYPERELASTIC_RUBBER

This is Material Type 77. This material model provides a general hyperelastic rubber model combined optionally with linear viscoelasticity as outlined by [Christensen 1980].

Card Format

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	PR	N				
Type	I	F	F	I				

Card 2 if N > 0, a least squares fit is computed from unixial data

Card Format

Card 2	1	2	3	4	5	6	7	8
Variable	SGL	SW	ST	LCID				
Type	F	F	F	F				

Card 2 if N = 0 define the following constants

Card Format

Card 2	1	2	3	4	5	6	7	8
Variable	C10	C01	C11	C20	C02			
Type	F	F	F	F	F			

Card Format for Viscoelastic Constants. Up to 6 cards may be input. A keyword card (with a "*" in column 1) terminates this input if less than 6 cards are used.

Optional Cards	1	2	3	4	5	6	7	8
Variable	GI	BETAI						
Type	F	F						

VARIABLE	DESCRIPTION
MID	Material identification. A unique number has to be chosen.
RO	Mass density
PR	Poissons ratio (>.49 is recommended, smaller values may not work and should not be used).
N	order of fit, (currently <3)

if N>0 test information from a uniaxial test are voided:

SGL	Specimen gauge length
SW	Specimen width
ST	Specimen thickness
LCID	Load curve ID giving the force versus actual change in the gauge length

If N=0, the following constants have to be defined:

C10	C_{10}
C01	C_{01}
C11	C_{11}
C20	C_{20}
C02	C_{02}
GI	Optional shear relaxation modulus for the ith term
BETAI	Optional decay constant if ith term

Rubber is generally considered to be fully incompressible since the bulk modulus greatly exceeds the shear modulus in magnitude. To model the rubber as an unconstrained material a hydrostatic work term, $W_H(J)$, is included in the strain energy functional which is function of the relative volume, *J*, [Ogden, 1984]:

$$W(J_1, J_2, J) = \sum_{p,q=0}^{n} C_{pq}(J_1 - 3)^p (J_2 - 3)^q + W_H(J)$$

$$J_1 = I_1 J^{-1/3}$$

$$J_2 = I_2 J^{-2/3}$$

In order to prevent volumetric work from contributing to the hydrostatic work the first and second invarients are modified as shown. This procedure is described in more detail by Sussman and Bathe [1987].

Rate effects are taken into account through linear viscoelasticity by a convolution integral of the form:

$$\sigma_{ij} = \int_0^t g_{ijkl}(t-\tau)\frac{\partial \varepsilon_{kl}}{\partial \tau} d\tau$$

or in terms of the second Piola-Kirchhoff stress, S_{ij}, and Green's strain tensor, E_{ij},

$$S_{ij} = \int_0^t G_{ijkl}(t-\tau)\frac{\partial E_{kl}}{\partial \tau} d\tau$$

where $g_{ijkl}(t-\tau)$ and $G_{ijkl}(t-\tau)$ are the relaxation functions for the different stress measures. This stress is addedto the stress tensor determined from the strain energy functional.

If we wish to include only simple rate effects, the relaxation function is represented by six terms from the Prony series:

$$g(t) = \alpha_0 + \sum_{m=1}^{N} \alpha_m e^{-\beta t}$$

given by,

$$g(t) = \sum_{i=1}^{n} G_i e^{-\beta_i t}$$

This model is effectively a Maxwell fluid which consists of a dampers and springs in series. We characterize this in the input by shear modulii, G_i, and decay constants, β_i. The viscoelastic behavior is optional and an arbitrary number of terms may be used.

The Mooney-Rivlin rubber model (model 27) is obtained by specifying n=1. In spite of the differences in formulations with Model 27, we find that the results obtained with this model are nearly identical with those of Material 27 as long as large values of Poisson's ratio are used.

*MAT_OGDEN_RUBBER

This is also Material Type 77. This material model provides the Ogden [1984] rubber model combined optionally with linear viscoelasticity as outlined by [Christensen 1980].

Card Format

Card 1

1	2	3	4	5	6	7	8
Variable	MID	RO	PR					
Type	I	F	F					

Cards 2 and 3. Define the following constants for the Ogden model.

Card Format

Card 2

1	2	3	4	5	6	7	8
Variable	MU1	MU2	MU3	MU4	MU5	MU6	MU7	MU8
Type	F	F	F	F	F	F	F	F

Card 3

1	2	3	4	5	6	7	8
Variable	ALPHA1	ALPHA2	ALPHA3	ALPHA4	ALPHA5	ALPHA6	ALPHA7	ALPHA8
Type	F	F	F	F	F	F	F	F

Card Format for Viscoelastic Constants. Up to 6 cards may be input. A keyword card (with a "*" in column 1) terminates this input if less than 6 cards are used.

Optional Cards	1	2	3	4	5	6	7	8
Variable	GI	BETAI						
Type	F	F						

VARIABLE	DESCRIPTION
MID	Material identification. A unique number has to be chosen.
RO	Mass density
PR	Poissons ratio (≥.49 is recommended, smaller values may not work and should not be used).
MUi	μ_i, the ith shear modulus, i varies up to 8. See discussion below.
ALPHAi	α_i, the ith exponent, i varies up to 8. See discussion below.
GI	Optional shear relaxation modulus for the ith term
BETAI	Optional decay constant if ith term

Rubber is generally considered to be fully incompressible since the bulk modulus greatly exceeds the shear modulus in magnitude. To model the rubber as an unconstrained material a hydrostatic work term is included in the strain energy functional which is function of the relative volume, *J*, [Ogden, 1984]:

$$W^* = \sum_{i=1}^{3}\sum_{j=1}^{n}\frac{\mu_j}{\alpha_j}\left(\lambda_i^{*\alpha_j} - 1\right) + \frac{1}{2}K(J-1)^2$$

The asterisk (*) indicates that the volumetric effects have be eliminated from the principal stretches, λ^*_j.. The number of terms, n, is may vary between 1 to 8 inclusive, and K is the bulk modulus.

Rate effects are taken into account through linear viscoelasticity by a convolution integral of the form:

$$\sigma_{ij} = \int_0^t g_{ijkl}(t-\tau)\frac{\partial \varepsilon_{kl}}{\partial \tau} d\tau$$

or in terms of the second Piola-Kirchhoff stress, S_{ij}, and Green's strain tensor, E_{ij},

$$S_{ij} = \int_0^t G_{ijkl}(t-\tau)\frac{\partial E_{kl}}{\partial \tau} d\tau$$

where $g_{ijkl}(t-\tau)$ and $G_{ijkl}(t-\tau)$ are the relaxation functions for the different stress measures. This stress is addedto the stress tensor determined from the strain energy functional.

If we wish to include only simple rate effects, the relaxation function is represented by six terms from the Prony series:

$$g(t) = \alpha_0 + \sum_{m=1}^{N} \alpha_m e^{-\beta t}$$

given by,

$$g(t) = \sum_{i=1}^{n} G_i e^{-\beta_i t}$$

This model is effectively a Maxwell fluid which consists of a dampers and springs in series. We characterize this in the input by shear modulii, G_i, and decay constants, β_i. The viscoelastic behavior is optional and an arbitrary number of terms may be used.

The Mooney-Rivlin rubber model (model 27) is obtained by specifying n=1. In spite of the differences in formulations with Model 27, we find that the results obtained with this model are nearly identical with those of Material 27 as long as large values of Poisson's ratio are used.

***MAT_SOIL_CONCRETE**

This is Material Type 78. This model permits concrete and soil to be efficiently modelled. See the explanations below.

Card Format

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	G	K	LCPV	LCYP	LCFP	LCRP
Type	I	F	F	F	F	F	F	F

Card 2	1	2	3	4	5	6	7	8
Variable	PC	OUT	B	FAIL				
Type	F	F	F	F				

VARIABLE	DESCRIPTION
MID	Material identification. A unique number has to be chosen.
RO	Mass density
G	Shear modulus
K	Bulk modulus
LCPV	Load curve ID for pressure versus volumetric strain. The pressure versus volumetric strain curve is defined for compression only. The sign convention requires that both pressure and compressive strain be defined as positive values where the compressive strain is taken as the negative value of the natural logrithm of the relative volume.
LCYP	Load curve ID for yield versus pressure: GT.0: von Mises stress versus pressure, LT.0: Second stress invariant, J_2, versus pressure.
LCFP	Load curve ID for plastic strain at which fracture begins versus pressure.

VARIABLE	DESCRIPTION
LCRP	Load curve ID for plastic strain at which residual strength is reached versus pressure.
PC	Pressure cutoff for tensile fracture
OUT	Output option for plastic strain in database: EQ.0: volumetric plastic strain, EQ.1: deviatoric plastic strain.
B	Residual strength factor after cracking, see Figure 19.21.
FAIL	Flag for failure: EQ.0: no failure, EQ.1: failure, when pressure reaches failure pressure element loses it ability to carry tension.

Pressure is positive in compression. Volumetric strain is defined as the natural log of the relative volume and is *positive* in compression where the relative volume, V, is the ratio of the current volume to the initial volume. The tabulated data should be given in order of increasing compression. If the pressure drops below the cutoff value specified, it is reset to that value and the deviatoric stress state is eliminated.

If the load curve ID (LCYP) is provided as a positive number, the deviatoric, perfectly plastic, pressure dependent, yield function ϕ, is given as

$$\phi = \sqrt{3J_2} - F(p) = \sigma_y - F(p)$$

where , *F(p)* is a tabulated function of yield stress versus pressure, and the second invarient, J_2, is defined in terms of the deviatoric stress tensor as:

$$J_2 = \frac{1}{2} S_{ij} S_{ij}$$

assuming that . If the ID is given as negative then the yield function becomes:

$$\phi = J_2 - F(p)$$

being the deviatoric stress tensor.

If cracking is invoked by setting the residual strength factor on card 2 to a value between 0.0 and 1.0, the yield stress is multiplied by a factor f which reduces with plastic strain according to a trilinear law as shown in Figure 19.21.

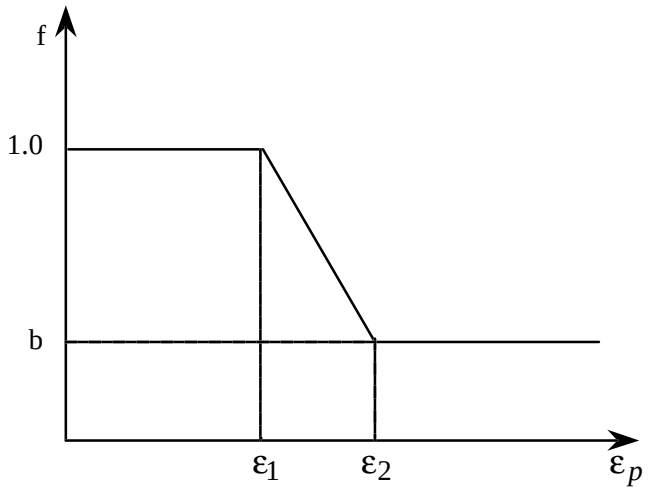


Figure 19.21. Strength reduction factor.

b = residual strength factor
ε_1 = plastic stain at which cracking begins.
ε_2 = plastic stain at which residual strength is reached.

ε_1 and ε_2 are tabulated function of pressure that are defined by load curves, see Figure 19.22. The values on the curves are pressure versus strain and should be entered in order of increasing pressure. The strain values should always increase monotonically with pressure.

By properly defining the load curves, it is possible to obtain the desired strength and ductility over a range of pressures, see Figure 19.23.

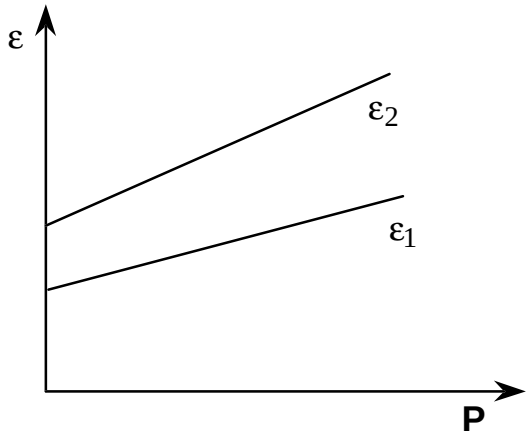


Figure 19.22. Cracking strain versus pressure.

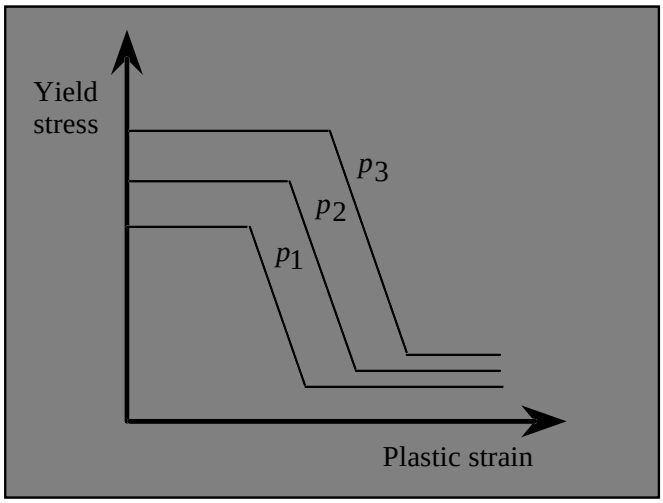


Figure 19.23.

***MAT_HYSTERETIC_SOIL**

This is Material Type 79. This model is a nested surface model with five superposed "layers" of elasto-perfectly plastic material, each with its own elastic modulii and yield values. Nested surface models give hysteric behavior, as the different "layers" yield at different stresses. See notes below.

Card Format

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	K0	P0	B	A0	A1	A2
Type	I	F	F	F	F	F	F	F

Card 2	1	2	3	4	5	6	7	8
Variable	DF	RP						
Type	F	F						

Card 3	1	2	3	4	5	6	7	8
Variable	GAM1	GAM2	GAM3	GAM4	GAM5			
Type	F	F	F	F	F			

Card 4	1	2	3	4	5	6	7	8
Variable	TAU1	TAU2	TAU3	TAU4	TAU5			
Type	F	F	F	F	F			

VARIABLE	DESCRIPTION
MID	Material identification. A unique number has to be chosen.
RO	Mass density
K0	Bulk modulus at the reference pressure
P0	Cut-off/datum pressure (must be 0≤ i.e. tensile). Below this pressure, stiffness and strength disappears; this is also the "zero" pressure for pressure-varying properties.
B	Exponent for pressure-sensitive moduli, b: $G = G_0(p - p_o)^b$ $K = K_0(p - p_o)^b$. b, must lie in the range 0≤b<1. Values close to 1 are not recommended because the pressure becomes indeterminate.
A0	Yield function constant a_o (Default = 1.0), see Material Type 5.
A1	Yield function constant a_1 (Default = 0.0), see Material Type 5.
A2	Yield function constant a_2 (Default = 0.0), see Material Type 5.
DF	Damping factor. Must be in the range 0≤df≤1: EQ.0: no damping, EQ.1: maximum damping.
RP	Reference pressure for following input data.
GAM1	γ_1, shear strain
GAM2	γ_2, shear strain
GAM3	γ_3, shear strain
GAM4	γ_4, shear strain
GAM5	γ_5, shear strain

VARIABLE	DESCRIPTION
TAU1	τ_1, shear stress at γ_1
TAU2	τ_2, shear stress at γ_2
TAU3	τ_3, shear stress at γ_3
TAU4	τ_4, shear stress at γ_4
TAU5	τ_5, shear stress at γ_5

The constants a_0, a_1, a_2 govern the pressure sensitivity of the yield stress. Only the ratios between these values are important - the absolute stress values are take from the stress-strain curve.

The stress strain pairs (γ_1, τ_1), ... (γ_5, τ_5) define a shear stress versus shear strain curve. The first point on the curve is assumed by default to be (0,0) and does not need to be entered. The slope of the curve must decrease with increasing γ. Not all five points need be to be defined. This curves applies at the reference pressure; at other pressures the curve varies according to a_0, a_1, and a_2 as in the soil and crushable foam model, Material 5, SOIL_AND_FOAM.

The elastic moduli G and K are pressure sensitive.

$$G = G_0\left(p - p_o\right)^b$$
$$K = K_0\left(p - p_o\right)^b$$

where G_0 and K_0 are the input values, p is the current pressure, p_0 the cut-off or reference pressure (must be zero or negative). If p attempts to fall below p_0 (i.e., more tensile) the shear stresses are set to zero and the pressure is set to p_0. Thus, the material has no stiffness or strength in tension. The pressure in compression is calculated as follows:

$$p = \left[-K_0 \ln\left(V\right)\right]^{1/1-b}$$

where V is the relative volume, i.e., the ratio between the original and current volume.

*MAT_PLASTICITY_WITH_DAMAGE

This is Material Type 81. An elasto-plastic material with an arbitrary stress versus strain curve and arbitrary strain rate dependency can be defined. Damage is considered before rupture occurs. Also, failure based on a plastic strain or a minimum time step size can be defined.

Card Format

Card 1

Variable	MID	RO	E	PR	SIGY	ETAN	EPPF	TDEL
Type	I	F	F	F	F	F	F	F
Default	none	none	none	none	none	0.0	10.E+20	10.E+20

Card 2

Variable	C	P	LCSS	LCSR	EPPFR			
Type	F	F	F	F	F			
Default	0	0	0	0	0			

Card 3

Variable	EPS1	EPS2	EPS3	EPS4	EPS5	EPS6	EPS7	EPS8
Type	F	F	F	F	F	F	F	F
Default	0	0	0	0	0	0	0	0

Card 4

Variable	ES1	ES2	ES3	ES4	ES5	ES6	ES7	ES8
Type	F	F	F	F	F	F	F	F
Default	0	0	0	0	0	0	0	0

VARIABLE	DESCRIPTION
MID	Material identification. A unique number has to be chosen.
RO	Mass density.
E	Young's modulus.
PR	Poisson's ratio.
SIGY	Yield stress.
ETAN	Tangent modulus, ignored if (LCSS.GT.0) is defined.
EPPF	Plastic strain at which material softening begins (logrithmic).
TDEL	Minimum time step size for automatic element deletion.
C	Strain rate parameter, C, see formula below.
P	Strain rate parameter, P, see formula below.
LCSS	Load curve ID. Load curve ID defining effective stress versus effective plastic strain. If defined EPS1-EPS8 and ES1-ES8 are ignored.
LCSR	Load curve ID defining strain rate scaling effect on yield stress.
EPPFR	Plastic strain at which material ruptures (logrithmic).
EPS1-EPS8	Effective plastic strain values (optional if SIGY is defined). At least 2 points should be defined.
ES1-ES8	Corresponding yield stress values to EPS1 - EPS8.

The stress strain behavior may be treated by a bilinear stress strain curve by defining the tangent modulus, ETAN. Alternately, a curve similar to that shown in Figure 19.4 is expected to be defined by (EPS1,ES1) - (EPS8,ES8); however, an effective stress versus effective plastic strain curve (LCSS) may be input instead if eight points are insufficient. The cost is roughly the same for either approach. The most general approach is to use the table definition (LCSS) discussed below.

Two options to account for strain rate effects are possible.

I. Strain rate may be accounted for using the Cowper and Symonds model which scales the yield stress with the factor

$$1+\left(\frac{\dot{\varepsilon}}{C}\right)^{1/p}$$

where $\dot{\varepsilon}$ is the strain rate. $\dot{\varepsilon}=\sqrt{\dot{\varepsilon}_{ij}}\dot{\varepsilon}_{ij}$

II. For complete generality a load curve (LCSR) to scale the yield stress may be input instead. In this curve the scale factor versus strain rate is defined.

*MAT_ORTHOTROPIC_VISCOELASTIC

This is Material Type 86. It allows the definition of an orthotropic material with a viscoelastic part. This model applies to shell elements.

Card Format

Card 1

Variable	MID	RO	EA	EB	EC	VF	K	
Type	I	F	F	F	F	F	F	

Card 2

Variable	G0	GINF	BETA	PRBA	PRCA	PRCB		
Type	F	F	F	F	F	F		

Card 3

Variable	GAB	GBC	GCA	AOPT	MANGLE			
Type	F	F	F	F	F			

Card 4

Variable	XP	YP	ZP	A1	A2	A3		
Type	F	F	F	F	F	F		

Card 5

Variable	V1	V2	V3	D1	D2	D3		
Type	F	F	F	F	F	F		

VARIABLE	DESCRIPTION
MID	Material identification. A unique number has to be chosen.
RO	Mass density
EA	Young's Modulus E_a
EB	Young's Modulus E_b
EC	Young's Modulus E_c
VF	Volume fraction of viscoelastic material
K	Elastic bulk modulus
G0	G_0, short-time shear modulus
GINF	G_∞, long-time shear modulus
BETA	β, decay constant
PRBA	Poisson's ratio, ν_{ba}
PRCA	Poisson's ratio, ν_{ca}
PRCB	Poisson's ratio, ν_{cb}
GAB	Shear modulus, G_{ab}
GBC	Shear modulus, G_{bc}
GCA	Shear modulus, G_{ca}

VARIABLE	DESCRIPTION
AOPT	Material axes option, see Figure 19.1: EQ. 0.0: locally orthotropic with material axes determined by element nodes as shown in Figure 19.1. Nodes 1, 2, and 4 of an element are identical to the Nodes used for the definition of a coordinate system as by *DEFINE_COORDINATE_NODES. EQ. 1.0: locally orthotropic with material axes determined by a point in space and the global location of the element center, this is the a-direction. EQ. 2.0: globally orthotropic with material axes determined by vectors defined below, as with *DEFINE_COORDINATE_VECTOR. EQ. 3.0: applicable to shell elements only. This option determines locally orthotropic material axes by offsetting the material axes by an angle to be specified from a line in the plane of the shell determined by taking the cross product of the vector **v** defined below with the shell normal vector.
MANGLE	Material angle, may be overwritten on the element card AOPT = 3.
XP YP ZP	Define coordinates of point p for AOPT = 1 and 4.
A1 A2 A3	Define components of vector a for AOPT = 2.
V1 V2 V3	Define components of vector v for AOPT = 3 and 4.
D1 D2 D3	Define components of vector d for AOPT = 2.

For the orthotropic definition it is referred to Material Type 2 and 21.

*MAT_CELLULAR_RUBBER

This is Material Type 87. This material model provides a cellular rubber model with confined air pressure combined with linear viscoelasticity as outlined by Christensen [1980]. See Figure 19.24.

Card Format

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	PR	N				
Type	I	F	F	I				

Card 2 if N > 0, a least squares fit is computed from unixial data

Card Format

Card 2	1	2	3	4	5	6	7	8
Variable	SGL	SW	ST	LCID				
Type	F	F	F	F				

Card 2 if N = 0, define the following constants

Card Format

Card 2	1	2	3	4	5	6	7	8
Variable	C10	C01	C11	C20	C02			
Type	F	F	F	F	F			

Card Format

Card 3	1	2	3	4	5	6	7	8
Variable	P0	PHI	IVS	G	BETA			
Type	I	F	F	F	F			

VARIABLE	DESCRIPTION
MID	Material identification. A unique number has to be chosen.
RO	Mass density
PR	Poisson's ratio, typical values are between .0 to .2. Due to the large compressibility of air, large values of Poisson's ratio generates physically meaningless results.
N	Order of fit (currently < 3). If n>0 then a least square fit is computed with uniaxial data. The parameters given on card 2 should be specified. Also see *MAT_MOONEY_RIVLIN_RUBBER (material model 27). A Poisson's ratio of .5 is assumed for the void free rubber during the fit. The Poisson's ratio defined on Card 1 is for the cellular rubber. A void fraction formulation is used.

Define, if N > 0:

SGL	Specimen gauge length l_0
SW	Specimen width
ST	Specimen thickness
LCID	Load curve ID giving the force versus actual change ΔL in the gauge length.

Define, if N = 0:

C10	Coefficient, C_{10}
C01	Coefficient, C_{01}
C11	Coefficient, C_{11}
C20	Coefficient, C_{20}
C02	Coefficient, C_{02}

VARIABLE	DESCRIPTION
P0	Initial air pressure, P_0
PHI	Ratio of cellular rubber to rubber density, Φ
IVS	Initial volumetric strain, γ_0
G	Optional shear relaxation modulus, *G*, for rate effects (viscosity)
BETA	Optional decay constant, β_1

Rubber is generally considered to be fully incompressible since the bulk modulus greatly exceeds the shear modulus in magnitude. To model the rubber as an unconstrained material a hydrostatic work term, $W_H(J)$, is included in the strain energy functional which is function of the relative volume, *J*, [Ogden, 1984]:

$$W(J_1, J_2, J) = \sum_{p,q=0}^{n} C_{pq}(J_1 - 3)^p (J_2 - 3)^q + W_H(J)$$

$$J_1 = I_1 I_3^{-1/3}$$

$$J_2 = I_2 I_3^{-2/3}$$

In order to prevent volumetric work from contributing to the hydrostatic work the first and second invarients are modified as shown. This procedure is described in more detail by Sussman and Bathe [1987].

The effects of confined air pressure in its overall response characteristics is included by augmenting the stress state within the element by the air pressure.

$$\sigma_{ij} = \sigma_{ij}^{sk} - \delta_{ij}\sigma^{air}$$

where σ_{ij}^{sk} is the bulk skeletal stress and σ^{air} is the air pressure computed from the equation:

$$\sigma^{air} = -\frac{p_0\gamma}{1+\gamma-\phi}$$

where p_0 is the initial foam pressure usually taken as the atmospheric pressure and γ defines the volumetric strain

$$\gamma = V - 1 + \gamma_0$$

where V is the relative volume of the voids and γ_0 is the initial volumetric strain which is typically zero. The rubber skeletal material is assumed to be incompressible.

Rate effects are taken into account through linear viscoelasticity by a convolution integral of the form:

$$\sigma_{ij} = \int_0^t g_{ijkl}(t-\tau)\frac{\partial \varepsilon_{kl}}{\partial \tau}\, d\tau$$

or in terms of the second Piola-Kirchhoff stress, S_{ij}, and Green's strain tensor, E_{ij},

$$S_{ij} = \int_0^t G_{ijkl}(t-\tau)\frac{\partial E_{kl}}{\partial \tau}\, d\tau$$

where $g_{ijkl}(t-\tau)$ and $G_{ijkl}(t-\tau)$ are the relaxation functions for the different stress measures. This stress is addedto the stress tensor determined from the strain energy functional.

Since we wish to include only simple rate effects, the relaxation function is represented by one term from the Prony series:

$$g(t) = \alpha_0 + \sum_{m=1}^{N} \alpha_m e^{-\beta t}$$

given by,

$$g(t) = E_d e^{-\beta_1 t}$$

This model is effectively a Maxwell fluid which consists of a damper and spring in series. We characterize this in the input by a shear modulus, G, and decay constant, β_1.

The Mooney-Rivlin rubber model (model 27) is obtained by specifying n=1 without air pressure and viscosity. In spite of the differences in formulations with Model 27, we find that the results obtained with this model are nearly identical with those of material type 27 as long as large values of Poisson's ratio are used.

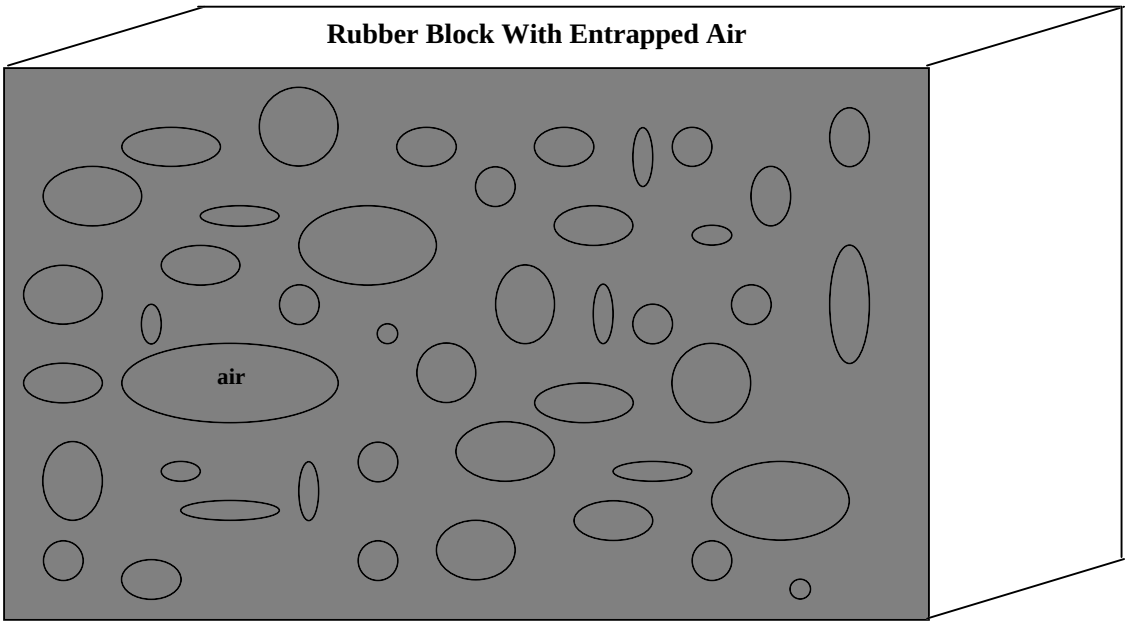


Figure 19.24. Cellular rubber with entrapped air. By setting the initial air pressure to zero, an open cell, cellular rubber can be simulated.

*MAT_ACOUSTIC

This is Material Type 90. This model is appropiate for tracking low pressure stress waves in an acoustic media such as air or water and can be used only with the acoustic pressure element formulation. The acoustic pressure element requires only one unknown per node. This element is very cost effective. Optionally, cavitation can be allowed.

Card Format

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	C	BETA	CF	ATMOS	GRAV	
Type	I	F	F	F	F	F	F	

Card 2	1	2	3	4	5	6	7	8
Variable	XP	YP	ZP	XN	YN	ZN		
Type	F	F	F	F	F	F		

VARIABLE	DESCRIPTION
MID	Material identification. A unique number has to be chosen.
RO	Mass density
C	Sound speed
BETA	Damping factor. Recommend values are between 0.1 and 1.0.
CF	Cavitation flag: EQ.0.0: off, EQ.1.0: on.
ATMOS	Atmospheric pressure (optional)
GRAV	Gravitational acceleration constant (optional)
XP	x-coordinate of free surface point

VARIABLE	DESCRIPTION
XP	y-coordinate of free surface point
YP	z-coordinate of free surface point
XN	x-direction cosine of free surface normal vector
YN	y-direction cosine of free surface normal vector
ZN	z-direction cosine of free surface normal vector

***MAT_SPRING_ELASTIC**

This allows to simulate a translational or rotational elastic spring located between two nodes. Only one degree of freedom is then connected.

Card Format

Card 1	1	2	3	4	5	6	7	8
Variable	MID	K						
Type	I	F						

VARIABLE	DESCRIPTION
MID	Material ID. A unique number has to be chosen.
K	Elastic stiffness (force/displacement) or (moment/rotation).

*MAT_DAMPER_VISCOUS

This material allows to simulate a linear translational or rotational damper located between two nodes. Only one degree of freedom is then connected.

Card Format

Card 1	1	2	3	4	5	6	7	8
Variable	MID	DC						
Type	I	F						

VARIABLE	DESCRIPTION
MID	Material ID. A unique number has to be chosen.
DC	Damping constant (force/displacement rate) or (moment/roation rate).

***MAT_SPRING_ELASTOPLASTIC**

This material allows to simulate an elastoplastic translational or rotational spring with isotropic hardening located between two nodes. Only one degree of freedom is then connected.

Card Format

Card 1	1	2	3	4	5	6	7	8
Variable	MID	K	KT	FY				
Type	I	F	F	F				

VARIABLE	DESCRIPTION
MID	Material number. A unique number has to be chosen.
K	Elastic stiffness (force/displacement) or (moment/rotation).
KT	Tangent stiffness (force/displacement) or (moment/rotation).
FY	Yield (force) or (moment).

*MAT_SPRING_NONLINEAR_ELASTIC

This material allows to simulate a nonlinear elastic translational or rotational spring with arbitrary force/displacement resp. moment/rotation dependency. Optionally, strain rate effects can be considered through a velocity dependent scale factor. With the spring located between two nodes, only one degree of freedom is connected.

Card Format

Card 1	1	2	3	4	5	6	7	8
Variable	MID	LCD	LCR					
Type	I	I	I					

VARIABLE	DESCRIPTION
MID	Material number. A unique numbe has to be chosen.
LCD	Load curve ID describing force versus displacement or moment versus rotation relationship
LCR	Optional loadcurve describing scale factor on force or moment as a function of relative velocity resp. rotational velocity.

*MAT_DAMPER_NONLINEAR_VISCOUS

This material allows to simulate a nonlinear versus translation or rotational damper with arbitrary force/velocity resp. moment/rotational velocity dependency. With the damper located between two nodes, only one degree of freedom is connected.

Card Format

Card 1	1	2	3	4	5	6	7	8
Variable	MID	LCDR						
Type	I	I						

VARIABLE	DESCRIPTION
MID	Material identification. A unique number has to be chosen.
LCDR	Load curve identification describing force versus rate-of-displacement relationship/resp. moment versus rate-of-rotation relationship.

*MAT_SPRING_GENERAL_NONLINEAR

This material allows to simulate a general nonlinear translational or rotational spring with arbitrary loading and unloading definitions. Optionally, hardening or softening can be defined. With the spring located between two nodes, only one degree of freedom is connected.

Card Format

Card 1	1	2	3	4	5	6	7	8
Variable	MID	LCDL	LCDU	BETA	TYI	CYI		
Type	I	I	I	F	F	F		

VARIABLE	DESCRIPTION
MID	Material identification. A unique number has to be chosen.
LCDL	Load curve identification describing force versus displacement resp. moment versus rotation relationship for loading, see Figure 19.25.
LCDU	Load curve identification describing force versus displacement resp. moment versus rotation relationship for unloading, see Figure 19.25.
BETA	Hardening parameter, β: EQ.0.0: tensile and compressive yield with strain softening (negative or zero slope allowed in the force versus disp. load curves), NE.0.0: kinematic hardening **without strain softening,** EQ.1.0: isotropic **hardening without strain softening.**
TYI	Initial yield force in tension (> 0)
CYI	Initial yield force in compression (< 0)

Load curve points are in the format (displacement, force or rotation, moment). The points must be in order starting with the most negative (compressive) displacement resp. rotation and ending with the most positive (tensile) value. The curves need not be symmetrical.

The displacement origin of the “unloading” curve is arbitrary, since it will be shifted as necessary as the element extends and contracts. On reverse yielding the “loading” curve will also be shifted along the displacement resp. rotation axis. The initial tensile and compressive yield forces (TYI and CYI) define a range within which the element remains elastic (i.e. the “loading” curve is

used for both loading and unloading). If at any time the force in the element exceeds this range, the element is deemed to have yielded, and at all subsequent times the “unloading” curve is used for unloading.

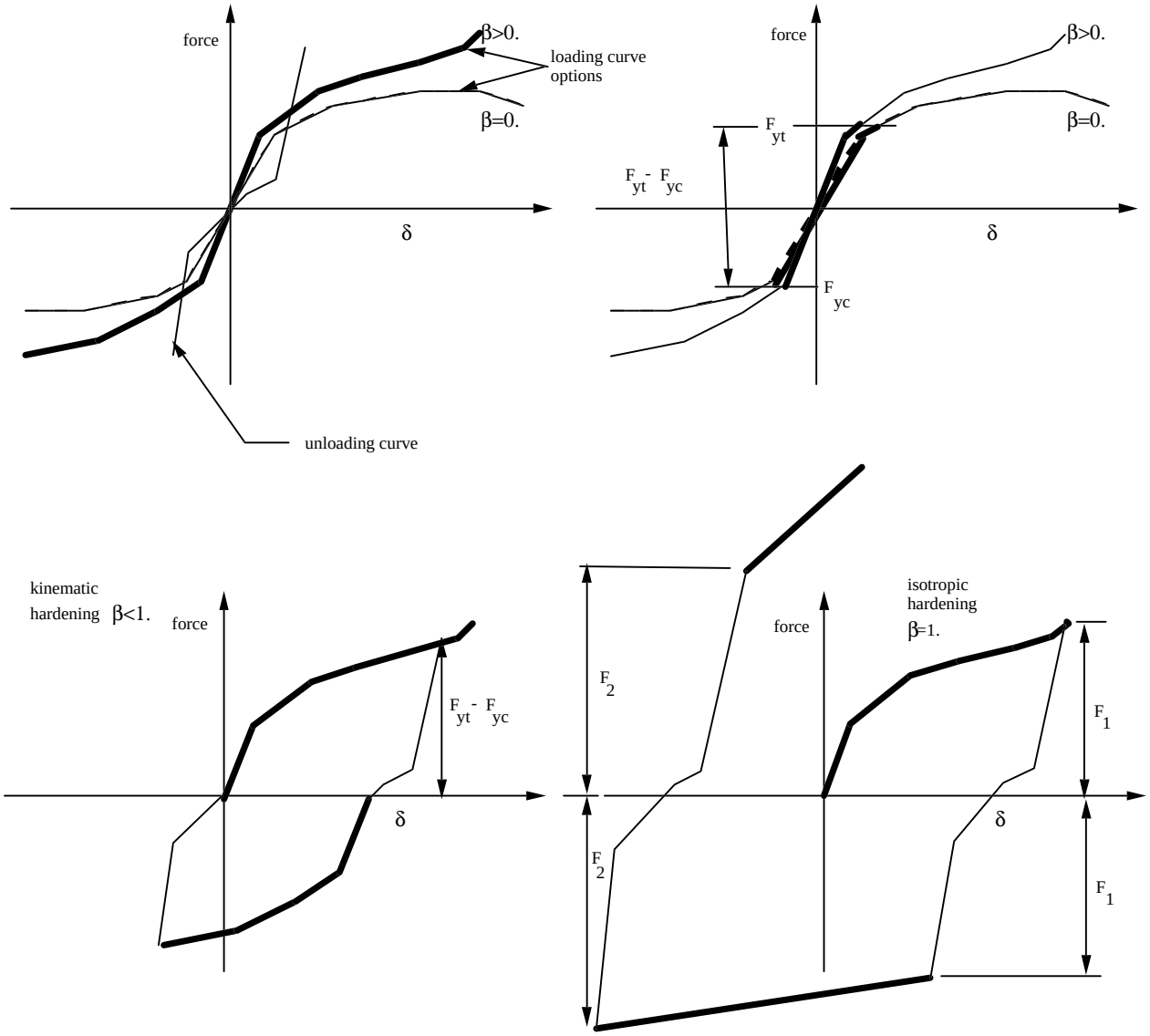


Figure 19.25. General nonlinear material for discrete elements.

***MAT_SPRING_MAXWELL**

This material allows to simulate a three Parameter Maxwell Viscoelastic translational or rotational spring. Optionally, a cutoff time with a remaining constant force/moment can be defined.

Card Format

Card 1	1	2	3	4	5	6	7	8
Variable	MID	K0	KI	BETA	TC	FC	COPT	
Type	I	F	F	F	F	F	F	
Default	---	---	---	---	10^{20}	0	0	

VARIABLE	DESCRIPTION
MID	Material identification. A unique number has to be chosen.
K0	K_0, short time stiffness
KI	K_∞, long time stiffness
BETA	Decay parameter.
TC	Cut off time. After this time a constant force/moment is transmitted.
FC	Force/moment after cutoff time
COPT	Time implementation option: EQ.0: incremental time change, NE.0: continuous time change.

The time varying stiffness K(t) may be described in terms of the input parameters as

$$K(t) = K_\infty + (K_0 - K_\infty)e^{-\beta t} \ .$$

This equation was implemented by Schwer [60] as either a continuous function of time or incrementally following the approach of Herrmann and Peterson [61]. The continous function of time implementation has the disadvantage of the energy absorber's resistance decaying with increasing time even without deformation. The advantage of the incremental implementation is that

an energy absorber must undergo some deformation before its resistance decays, i.e., there is no decay until impact, even in delayed impacts. The disadvantage of the incremental implementation is that very rapid decreases in resistance cannot be easily matched.

***MAT_SPRING_INELASTIC**

This material allows to simulate an onelastic tension or compression only, translational or rotational spring. Optionally, a user-specified unloading stiffness can be taken instead of the maximum loading stiffness.

Card Format

Card 1	1	2	3	4	5	6	7	8
Variable	MID	LCFD	KU	CTF				
Type	I	I	F	F				

VARIABLE	DESCRIPTION
MID	Material identification. A uniques number has to be chosen.
LCFD	Load curve identification describing arbitrary force/torque versus displacement/twist relationship. This curve must be defined in the positive force-displacement quadrant regardless of whether the spring acts in tension or compression.
KU	Unloading stiffness (optional). If zero, the maximum loading stiffness in the force/displacement resp. moment/twist curve is used.
CTF	Flag for compression/tension: EQ.-1.0: tension only, EQ.0.0: default is set to 1.0, EQ.1.0: compression only.

*MAT_SEATBELT

Purpose: Define seat belt material. See notes below.

Card Format

Card 1	1	2	3	4	5	6	7	8
Variable	MID	MPUL	LLCID	ULCID	LMIN			
Type	I	F	I	I	F			
Default	0	0.	0	0	0.0			

VARIABLE	DESCRIPTION
MID	Belt material number. A unique number has to be chosen.
MPUL	Mass per unit length
LLCID	Load curve identification for loading (strain/force with engineering strain).
ULCID	Load curve identification for unloading (strain/force with engineering strain).
LMIN	Minimum length (for elements connected to slip rings and retractors), see notes below.

Each belt material defines stretch characteristics and mass properties for a set of belt elements. The user enters a load curve for loading, the points of which are (Strain, Force). Strain is defined as engineering strain, i.e.

$$Strain = \frac{current\ length}{initial\ length} - 1 \ .$$

Another similar curve is entered to describe the unloading behavior. Both loadcurves should start at the origin (0,0) and contain positive force and strain values only. The belt material is tension only with zero forces being generated whenever the strain becomes negative. The first non-zero point on the loading curve defines the initial yield point of the material. On unloading, the unloading curve is shifted along the strain axis until it crosses the loading curve at the 'yield' point from which

unloading commences. If the initial yield has not yet been exceeded or if the origin of the (shifted) unloading curve is at negative strain, the original loading curves will be used for both loading and unloading. If the strain is less than the strain at the origin of the unloading curve, the belt is slack and no force is generated. Otherwise, forces will then be determined by the unloading curve for unloading and reloading until the strain again exceeds yield after which the loading curves will again be used.

A small amount of damping is automatically included. This reduces high frequency oscillation, but, with realistic force-strain input characteristics and loading rates, does not significantly alter the overall forces-strain performance. The damping forced opposes the relative motion of the nodes and is limited by stability:

$$D = \frac{.1 \times mass \times relative\ velocity}{time\ step\ size}$$

In addition, the magnitude of the damping force is limited to one-tenth of the force calculated from the force-strain relationship and is zero when the belt is slack. Damping forces are not applied to elements attached to sliprings and retractors.

The user inputs a mass per unit length that is used to calculate nodal masses on initialization.

A ‘minimum length’ is also input. This controls the shortest length allowed in any element and determines when an element passes through sliprings or are absorbed into the retractors. One-tenth of a typical initial element length is usually a good choice.

***MAT_THERMAL_OPTION**

The *MAT_THERMAL_ cards allow thermal properties to be defined in coupled structural/thermal and thermal only analyses, see *CONTROL_SOLUTION. Thermal properties must be defined for all solid and shell elements in such analyses. Thermal properties need not be defined for beam or discrete elements as these elements are not accounted for in the thermal phase of the calculation. However dummy thermal properties will be echoed for these elements in the D3HSP file.

Thermal material properties are specified by a thermal material ID number (TMID), this number is independent of the material ID number (MID) defined on all other *MAT_.. property cards. In the same analysis identical TMID and MID numbers may exist. The TMID and MID numbers are related through the *PART card.

*MAT_THERMAL_ISOTROPIC

This is thermal material property type 1. It allows isotropic thermal properties to be defined.

Card Format (1 of 2)

	1	2	3	4	5	6	7	8
Variable	TMID	TRO	TGRLC	TGMULT				
Type	I	F	F	F				

Card Format (2 of 2)

	1	2	3	4	5	6	7	8
Variable	HC	TC						
Type	F	F						

VARIABLE	DESCRIPTION
TMID	Thermal material identification, a unique number has to be chosen.
TRO	Thermal density: EQ 0.0 default to structural density.
TGRLC	Thermal generation rate curve number, see *DEFINE_CURVE: GT.0: function versus time, EQ.0: use constant multiplier value, TGMULT, LT.0: function versus temperature.
TGMULT	Thermal generation rate multiplier: EQ.0.0: no heat generation.
HC	Heat capacity
TC	Thermal conductivity

*MAT_THERMAL_ORTHOTROPIC

This is thermal material property type 2. It allows orthotropic thermal properties to be defined.

Card Format (1 of 4)

	1	2	3	4	5	6	7	8
Variable	TMID	TRO	TGRLC	TGMULT	AOPT			
Type	I	F	F	F	F			

Card Format (2 of 4)

	1	2	3	4	5	6	7	8
Variable	HC	K1	K2	K3				
Type	F	F	F	F				

Card Format (3 of 4)

	1	2	3	4	5	6	7	8
Variable	XP	YP	ZP	A1	A2	A3		
Type	F	F	F	F	F	F		

Card Format (4 of 4)

	1	2	3	4	5	6	7	8
Variable	D1	D2	D3					
Type	F	F	F					

VARIABLE	DESCRIPTION
TMID	Thermal material identification, a unique number has to be chosen.
TRO	Thermal density: EQ 0.0 default to structural density.
TGRLC	Thermal generation rate curve number, see *DEFINE_CURVE: GT.0: function versus time, EQ.0: use constant multiplier value, TGMULT, LT.0: function versus temperature.
TGMULT	Thermal generation rate multiplier: EQ.0.0: no heat generation.
AOPT	Material axes definition: EQ.0: locally orthotropic with material axes by element nodes N_1, N_2 and N_4, EQ.1: locally orthotropic with material axes determined by a point in space and global location of element center, EQ.2: globally orthotropic with material axes determined by vectors.
HC	Heat capacity
K_1	Thermal conductivity K_1 in local x-direction
K_2	Thermal conductivity K_2 in local y-direction
K_3	Thermal conductivity K_3 in local z-direction
XP, YP, ZP	Define coordinate of point **p** for AOPT = 1
A1, A2, A3	Define components of vector **a** for AOPT = 2
D1, D2, D3	Define components of vector **v** for AOPT = 2

*MAT_THERMAL_ISOTROPIC_TD

This is thermal material property type 3. It allows temperture dependent isotropic properties to be defined. The temperature dependency is defined by specifying a minimum of two and a maximum of eight data points. The properties must be defined for the tempertaure range that the material will see in the analysis.

Card Format (1 of 4)

	1	2	3	4	5	6	7	8
Variable	TMID	TRO	TGRLC	TGMULT				
Type	I	F	F	F				

Card Format (2 of 4)

	1	2	3	4	5	6	7	8
Variable	T1	T2	T3	T4	T5	T6	T7	T8
Type	F	F	F	F	F	F	F	F

Card Format (3 of 4)

	1	2	3	4	5	6	7	8
Variable	C1	C2	C3	C4	C5	C6	C7	C8
Type	F	F	F	F	F	F	F	F

Card Format (4 of 4)

	1	2	3	4	5	6	7	8
Variable	K1	K2	K3	K4	K5	K6	K7	K8
Type	F	F	F	F	F	F	F	F

VARIABLE	DESCRIPTION
TMID	Thermal material identification, a unique number has to be chosen.
TRO	Thermal density: EQ 0.0 default to structural density.
TGRLC	Thermal generation rate curve number, see *DEFINE_CURVE: GT.0: function versus time, EQ.0: use constant multiplier value, TGMULT, LT.0: function versus temperature.
TGMULT	Thermal generation rate multiplier: EQ.0.0: no heat generation.
T1 ... T8	Temperatures (T1 ... T8)
C1 ... C8	Heat capacity at T1 ... T8
K1 ... K8	Thermal conductivity at T1 ... T8

*MAT_THERMAL_ORTHOTROPIC_TD

This is thermal material property type 4. It allows temperture dependent orthotropic properties to be defined. The temperature dependency is defined by specifying a minimum of two and a maximum of eight data points. The properties must be defined for the tempertaure range that the material will see in the analysis.

Card Format (1 of 8)

	1	2	3	4	5	6	7	8
Variable	TMID	TRO	TGRLC	TGMULT	AOPT			
Type	I	F	F	F	F			

Card Format (2 of 8)

	1	2	3	4	5	6	7	8
Variable	T1	T2	T3	T4	T5	T6	T7	T8
Type	F	F	F	F	F	F	F	F

Card Format (3 of 8)

	1	2	3	4	5	6	7	8
Variable	C1	C2	C3	C4	C5	C6	C7	C8
Type	F	F	F	F	F	F	F	F

Card Format (4 of 8)

	1	2	3	4	5	6	7	8
Variable	(K1) 1	(K1) 2	(K1) 3	(K1) 4	(K1) 5	(K1) 6	(K1) 7	(K1) 8
Type	F	F	F	F	F	F	F	F

Card Format (5 of 8)

	1	2	3	4	5	6	7	8
Variable	(K2) 1	(K2) 2	(K2) 3	(K2) 4	(K2) 5	(K2) 6	(K2) 7	(K2) 8
Type	F	F	F	F	F	F	F	F

Card Format (6 of 8)

	1	2	3	4	5	6	7	8
Variable	(K3) 1	(K3) 2	(K3) 3	(K3) 4	(K3) 5	(K3) 6	(K3) 7	(K3) 8
Type	F	F	F	F	F	F	F	F

Card Format (7 of 8)

	1	2	3	4	5	6	7	8
Variable	XP	YP	ZP	A1	A2	A3		
Type	F	F	F	F	F	F		

Card Format (8 of 8)

	1	2	3	4	5	6	7	8
Variable	D1	D2	D3					
Type	F	F	F					

VARIABLE	DESCRIPTION
TMID	Thermal material identification, a unique number has to be chosen.
TRO	Thermal density: EQ 0.0 default to structural density.
TGRLC	Thermal generation rate curve number, see *DEFINE_CURVE: GT.0: function versus time, EQ.0: use constant multiplier value, TGMULT, LT.0: function versus temperature.
TGMULT	Thermal generation rate multiplier: EQ.0.0: no heat generation.
AOPT	Material axes definition: EQ.0: locally orthotropic with material axes by element nodes N_1, N_2 and N_4, EQ.1: locally orthotropic with material axes determined by a point in space and global location of element center, EQ.2: globally orthotropic with material axes determined by vectors.
T1 ... T8	Temperatures (T1 ... T8)
C1 ... C8	Heat capacity at T1 ... T8
$(K1)_1$ ... $(K1)_8$	Thermal conductivity K_1 in local x-direction at T1 ... T8
$(K2)_1$ ... $(K2)_8$	Thermal conductivity K_2 in local y-direction at T1 ... T8
$(K3)_1$ ... $(K3)_8$	Thermal conductivity K_3 in local z-direction at T1 ... T8
XP, YP, ZP	Define coordinate of point **p** for AOPT = 1
A1, A2, A3	Define components of vector **a** for AOPT = 2
D1, D2, D3	Define components of vector **v** for AOPT = 2

***MAT_THERMAL_ISOTROPIC_PHASE_CHANGE**

This is thermal material property type 9. It allows temperture dependent isotropic properties with phase change to be defined. The latent heat of the material is defined together with the solidus and liquidus temperatures. The temperature dependency is defined by specifying a minimum of two and a maximum of eight data points. The properties must be defined for the tempertaure range that the material will see in the analysis.

Card Format (1 of 5)

	1	2	3	4	5	6	7	8
Variable	TMID	TRO	TGRLC	TGMULT				
Type	I	F	F	F				

Card Format (2 of 5)

	1	2	3	4	5	6	7	8
Variable	T1	T2	T3	T4	T5	T6	T7	T8
Type	F	F	F	F	F	F	F	F

Card Format (3 of 5)

	1	2	3	4	5	6	7	8
Variable	C1	C2	C3	C4	C5	C6	C7	C8
Type	F	F	F	F	F	F	F	F

Card Format (4 of 5)

	1	2	3	4	5	6	7	8
Variable	K1	K2	K3	K4	K5	K6	K7	K8
Type	F	F	F	F	F	F	F	F

Card Format (5 of 5)

	1	2	3	4	5	6	7	8
Variable	SOLT	LIQT	LH					
Type	F	F	F					

VARIABLE	DESCRIPTION
TMID	Thermal material identification, a unique number has to be chosen.
TRO	Thermal density: EQ 0.0 default to structural density.
TGRLC	Thermal generation rate curve number, see *DEFINE_CURVE: GT.0: function versus time, EQ.0: use constant multiplier value, TGMULT, LT.0: function versus temperature.
TGMULT	Thermal generation rate multiplier: EQ.0.0: no heat generation.
T1 ... T8	Temperatures (T1 ... T8)
C1 ... C8	Heat capacity at T1 ... T8
K1 ... K8	Thermal conductivity at T1 ... T8
SOLT	Solidus temperature, T_S (must be < T_L)
LIQT	Liquidus temperature, T_L (must be > T_S)
LH	Latent heat

During phase change, that is between the solidus and liquidus temperatures, the heat capacity of the material will be enhanced to account for the latent heat as follows:

$$c(t) = m\left[1 - \cos 2\pi\left(\frac{T - T_S}{T_L - T_S}\right)\right] \qquad T_S < T < T_L$$

Where

T_L = liquidus termperature

T_S = solidus termperature

T = termperature

m = multiplier such that $\lambda = \int_{T_S}^{T_L} C(T)dT$

λ = latent heat

c = hear capcity

*MAT_THERMAL_ISOTROPIC_TD_LC

This is thermal material property type 10. It allows isotropic thermal properties that are temperature dependent specified by load curves to be defined. The properties must be defined for the tempertaure range that the material will see in the analysis.

Card Format (1 of 2)

	1	2	3	4	5	6	7	8
Variable	TMID	TRO	TGRLC	TGMULT				
Type	I	F	F	F				

Card Format (2 of 2)

	1	2	3	4	5	6	7	8
Variable	HCLC	TCLC						
Type	F	F						

VARIABLE	DESCRIPTION
TMID	Thermal material identification, a unique number has to be chosen.
TRO	Thermal density: EQ 0.0 default to structural density.
TGRLC	Thermal generation rate curve number, see *DEFINE_CURVE: GT.0: function versus time, EQ.0: use constant multiplier value, TGMULT, LT.0: function versus temperature.
TGMULT	Thermal generation rate multiplier: EQ.0.0: no heat generation.
HCLC	Load curve ID specifying heat capacity vs. temperature.
TCLC	Load curve ID specifying thermal conductivity vs. temperature.

*NODE

***NODE**

Purpose: Define a node and its coordinates in the global coordinate system. Also, the boundary conditions in global directions can be specified. Generally, nodes are assigned to elements; however, exceptions are possible, see remark 2 below.

Card Format (I8,3E16.0,2I8)

Card 1	1	2	3	4	5	6	7	8	9	10
Variable	NID	X		Y		Z		TC	RC	
Type	I	F		F		F		F	F	
Default	none	0.		0.		0.		0.	0.	
Remarks								1	1	

VARIABLE	DESCRIPTION
NID	Node number
X	x coordinate
Y	y coordinate
Z	z coordinate
TC	Translational Constraint: EQ.0: no constraints, EQ.1: constrained x displacement, EQ.2: constrained y displacement, EQ.3: constrained z displacement, EQ.4: constrained x and y displacements, EQ.5: constrained y and z displacements, EQ.6: constrained z and x displacements, EQ.7: constrained x, y, and z displacements.

VARIABLE	DESCRIPTION
RC	Rotational constraint: EQ.0: no constraints, EQ.1: constrained x rotation, EQ.2: constrained y rotation, EQ.3: constrained z rotation, EQ.4: constrained x and y rotations, EQ.5: constrained y and z rotations, EQ.6: constrained z and x rotations, EQ.7: constrained x, y, and z rotations.

Remarks:

1. Boundary conditions can also be defined using options in the ***CONSTRAINT** section of the manual.

2. A node without an element or a mass attached to it will be assigned a very small amount of mass. Generally, massless nodes should not cause any problems but in rare cases may create stability problems if these massless nodes interact with the structure. Warning messages are printed when massless nodes are found. Also, massless nodes are used with rigid bodies to place joints, see *CONSTRAINED_EXTRA_NODES_OPTION and *CONSTRAINED_NODAL_RIGID_BODY.

*PART

***PART_OPTION**

Available options are:

<BLANK>
PART_INERTIA
PART_REPOSITION

Purpose: Define parts, i.e., combine material and section information as well as hourglass control, thermal properties, and general specification for adaptivity.

The INERTIA option allows the inertial properties and initial conditions to be defined rather than calculated from the finite element mesh. This applies to rigid bodies , see *MAT_RIGID, only. The REPOSITION option applies to deformable materials and is used to reposition deformable materials attached to rigid dummy components whose motion is controlled by either CAL3D or MADYMO. At the beginning of the calculation each component controlled by CAL3D/MADYMO is automatically repositioned to be consistent with the CAL3D/MADYMO input. However, deformable materials attached to these component will not be repositioned unless this option is used.

Card Format

Card 1

Variable	HEADING	
Type	C	
Default	none	
Remarks	1	

Card 2	1	2	3	4	5	6	7	8
Variable	PID	SECID	MID	EOSID	HGID	GRAV	ADPOPT	TMID
Type	I	I	I	I	I	I	I	I
Default	none	none	none	0	0	0	0	0

Additional Cards are required for the INERTIA option.

Card 3	1	2	3	4	5	6	7	8
Variable	XC	YC	ZC	TM	IRCS			
Type	F	F	F	F	I			

Card 4	1	2	3	4	5	6	7	8
Variable	IXX	IXY	IXZ	IYY	IYZ	IZZ		
Type	F	F	F	F	F	F		

Card 5	1	2	3	4	5	6	7	8
Variable	VTX	VTY	VTZ	VRX	VRY	VRZ		
Type	F	F	F	F	F	F		

Optional card required for the IRCS=1.

Card 6	1	2	3	4	5	6	7	8
Variable	XL	YL	ZL	XLIP	YLIP	ZLIP		
Type	F	F	F	F	F	F		
Remark	2	2	2	2	2	2		

Additional Card is required for the REPOSITION option.

Optional	1	2	3	4	5	6	7	8
Variable	CMSN	MDEP	MOVOPT					
Type	I	I	I					

VARIABLE	DESCRIPTION
HEADING	Heading for the part
PID	Part identification
SECID	Section identification defined in the *SECTION section
MID	Material identification defined in the *MAT section
EOSID	Equation of state identification defined in the *EOS section. Nonzero only for solid elements using a an equation of state to compute pressure.
HGID	Hourglass/bulk viscosity identification defined in the *HOURGLASS Section: EQ.0: default values are used.
GRAV	Part initialization for gravity loading: EQ.0: all parts initialized, EQ.1: only current material initialized.
ADPOPT	Indicate if this part is adapted or not. see also *CONTROL_ADAPTIVITY: EQ.0: no adaptivity, EQ.1: yes.

VARIABLE	DESCRIPTION
TMID	Thermal material property identification defined in the *MAT_THERMAL Section. Thermal properties must be specified for all solid, shell, and thick shell parts if a thermal or coupled thermal structural/analysis is being performed. Beams and discrete elements are not considered in thermal analyses. EQ.0: defaults to MID
XC	x-coordinate of center of mass
YC	y-coordinate of center of mass
ZC	z-coordinate of center of mass
TM	Translational mass
IRCS	Flag for inertia tensor reference coordinate system: EQ.0: global inertia tensor, EQ.1: principal moments of inertia with orientation vectors.
IXX	I_{xx}, xx component of inertia tensor
IXY	I_{xy} (set to zero if IRCS=1)
IXZ	I_{xz} (set to zero if IRCS=1)
IYY	I_{yy}, yy component of inertia tensor
IYZ	I_{yz} (set to zero if IRCS=1)
IZZ	I_{zz}, zz component of inertia tensor
VTX	initial translational velocity of rigid body in x direction
VTY	initial translational velocity of rigid body in y direction
VTZ	initial translational velocity of rigid body in z direction
VRX	initial rotational velocity of rigid body about x axis
VRY	initial rotational velocity of rigid body about y axis
VRZ	initial rotational velocity of rigid body about z axis
XL	x-coordinate of local x-axis. Origin lies at (0,0,0).
YL	y-coordinate of local x-axis
ZL	z-coordinate of local x-axis

VARIABLE	DESCRIPTION
XLIP	x-coordinate of local in-plane vector
YLIP	y-coordinate of local in-plane vector
ZLIP	z-coordinate of local in-plane vector
CMSN	CAL3D segment number/MADYMO system number. See the numbering in the corresponding program.
MDEP	MADYMO ellipse/plane number: GT.0: ellipse number, EQ.0: default, LT.0: absolute value is plane number.
MOVOPT	Flag to deactivate moving for merged rigid bodies, see *CONSTRAINED_ RIGID_BODIES. This option allows a merged rigid body to be fixed in space while the nodes and elements of the generated CAL3D/MADYMO parts are repositioned: EQ.0: merged rigid body is repositioned, EQ.1: merged rigid body is not repositioned.

Remarks:

1. HEADING default is standard material description, e.g. Material Type 1. In case of SMUG post processing place PSHELL (or PBAR, or PSOLID) in columns 1-8 and Property name in columns 34-41.

2. The local cartesian coordinate system is defined as described in *DEFINE_COORDINATE_ VECTOR. The local z-axis vector is the vector cross product of the x axis and the in plane vector. The local y-axis vector is finally computed as the vector cross product of the z-axis vector and the x-axis vector.

*RIGIDWALL

Two keywords are used in this section to define rigid surfaces:

*RIGIDWALL_GEOMETRIC_OPTION_{OPTION}
*RIGIDWALL_PLANAR_{OPTION}_{OPTION}_{OPTION}

*RIGIDWALL

***RIGIDWALL_GEOMETRIC_OPTION_OPTION**

Available forms include (one is mandatory):

RIGIDWALL_GEOMETRIC_FLAT
RIGIDWALL_GEOMETRIC_PRISM
RIGIDWALL_GEOMETRIC_CYLINDER
RIGIDWALL_GEOMETRIC_SPHERE

If prescribed motion is desired an additional option is available:

MOTION

One of the shape types **[FLAT, PRISM, CYLINDER, SPHERE]** must be specified, followed by the optional definition of **MOTION**, both on the same line with ***RIGIDWALL_ GEOMETRIC**

Purpose: Define a rigid wall with an analytically described form. Four forms are possible. A prescribed motion is optional. For general rigid bodies with arbitrary surfaces and motion, refer to the *CONTACT_ENTITY definition.

Card Format (1 of 3)

Card 1 1 2 3 4 5 6 7 8

Variable	NSID	NSIDEX	BOXID					
Type	I	I	I					
Default	none	0	0					
Remarks								

Card Format (2 of 3)

Card 2	1	2	3	4	5	6	7	8
Variable	XT	YT	ZT	XH	YH	ZH	FRIC	
Type	F	F	F	F	F	F	F	
Default	0.	0.	0.	0.	0.	0.	0.	
Remarks								

Card Format (3 of 3) (Required if FLAT is specified after the keyword)

A plane with a finite size or with an infinite size can be defined, see Figure 22.1. The vector **m** is computed as the vector cross product **n** X **l**. The origin, which is the tail of the normal vector, is the corner point of the finite size plane.

Card 3	1	2	3	4	5	6	7	8
Variable	XHEV	YHEV	ZHEV	LENL	LENM			
Type	F	F	F	F	F			
Default	0.	0.	0.	infinity	infinity			

Card Format (3 of 3) (Required if PRISM is specified after the keyword)

The description of the definition of a plane with finite size is enhanced by an additional length in the direction negative to **n**, see Figure 22.1.

Card 3	1	2	3	4	5	6	7	8
Variable	XHEV	YHEV	ZHEV	LENL	LENM	LENP		
Type	F	F	F	F	F	F		
Default	none	0.	0.	infinity	infinity	infinity		

Card Format (3 of 3) (Required if CYLINDER is specified after the keyword)

The tail of **n** specifies the top plane of the cylinder. The length is defined in the direction negative to **n**, see Figure 22.1.

Card 3	1	2	3	4	5	6	7	8
Variable	RADCYL	LENCYL						
Type	F	F						
Default	none	infinity						

Card Format (3 of 3) (Required if SPHERE is specified after the keyword)

The center of the sphere is identical to the tail of **n**, see Figure 22.1.

Card 3	1	2	3	4	5	6	7	8
Variable	RADSPH							
Type	F							
Default	0.							

Optional Card (Required if MOTION is specified after the keyword)

Optional	1	2	3	4	5	6	7	8
Variable	LCID	OPT	VX	VY	VZ			
Type	I	I	F	F	F			
Default	none	none	none	none	none			

VARIABLE	DESCRIPTION
NSID	Nodal set ID containing slave nodes, see *SET_NODE_OPTION: EQ.0: all nodes are slave to rigid wall.
NSIDEX	Nodal set ID containing nodes that exempted as slave nodes, see *SET_ NODE_OPTION.
BOXID	If defined, only nodes in box are included as slave nodes to rigid wall.
XT	x-coordinate of tail of any outward drawn normal vector, **n**, originating on wall (tail) and terminating in space (head), see Figure 22.1.
YT	y-coordinate of tail of normal vector **n**
ZT	z-coordinate of tail of normal vector **n**
XH	x-coordinate of head of normal vector **n**
YH	y-coordinate of head of normal vector **n**
ZH	z-coordinate of head of normal vector **n**
FRIC	Interface friction: EQ.0.0: frictionless sliding after contact, EQ.1.0: stick condition after contact, 0.<FRIC<1.: Coulomb friction coefficient.
XHEV	x-coordinate of head of edge vector **l,** see Figure 22.1.
YHEV	y-coordinate of head of edge vector **l**
ZHEV	z-coordinate of head of edge vector **l**
LENL	Length of **l** edge. A zero valure defines an infinite size plane.
LENM	Length of **m** edge. A zero valure defines an infinite size plane.
LENP	Length of prism in the direction negative to **n**, see Figure 22.1.
RADCYL	Radius of cylinder
LENCYL	Length of cylinder, see Figure 22.1. Only if a valure larger than zero is specified is a finite length assumed.
RADSPH	Radius of sphere
LCID	Stonewall motion curve number, see *DEFINE_CURVE.
OPT	Type of motion: EQ.0: velocity specified, EQ.1: displacement specified.

VARIABLE	DESCRIPTION
VX	x-direction cosine of velocity/displacement vector
VY	y-direction cosine of velocity/displacement vector
VZ	z-direction cosine of velocity/displacement vector

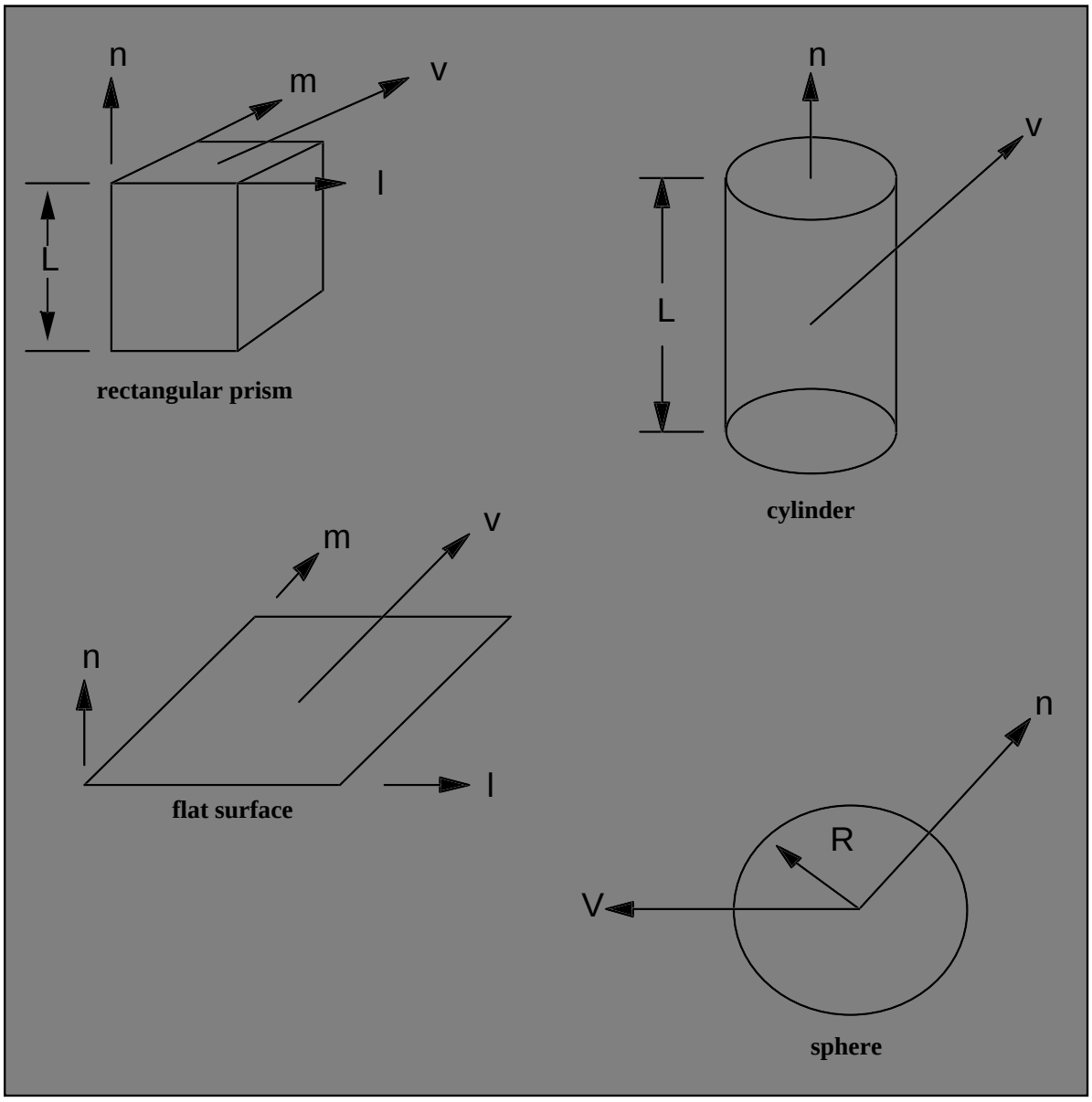


Figure 22.1. Vector **n** determines the orientation of the generalized stonewalls. For the prescribed motion options the wall can be moved in the direction **V** as shown.

*RIGIDWALL

***RIGIDWALL_PLANAR_OPTION_OPTION_OPTION**

Available options include:

ORTHO
FINITE
MOVING
FORCES

The ordering of the options in the input below must be observed but the ordering of the options on the command line is unimportant, i.e.; the **ORTHO** card is first, the **FINITE** definition card below must preceed the **MOVING** definition card, and the **FORCES** definition card should be last. The **ORTHO** option does not apply if the **MOVING** option is used.

Purpose: Define planar rigid walls with either finite or infinte size (**FINITE**). Orthotropic friction can be defined (**ORTHO**). Also, the plane can possess a mass and an initial velocity (**MOVING**); otherwise, the wall is assumed to be stationary. The **FORCES** option allows the specification of segments on the rigid walls on which the contact forces are computed. In order to achieve a more physical reaction related to the force versus time curve, the SOFT value on the **FORCES** card can be specified.

Card Format

Card 1	1	2	3	4	5	6	7	8
Variable	NSID	NSIDEX	BOXID					
Type	I	I	I					
Default	none	0	0					

Card 2	1	2	3	4	5	6	7	8
Variable	XT	YT	ZT	XH	YH	ZH	FRIC	
Type	F	F	F	F	F	F	F	
Default	0.	0.	0.	0.	0.	0.	0.	

Optional 2 Cards (Required if ORTHO is specified after the keyword). See Figure 22.2 for the definition of orthotropic friction.

	1	2	3	4	5	6	7	8
Variable	SFRICA	SFRICB	DFRICA	DFRICB	DECAYA	DECAYB		
Type	F	F	F	F	F	F		
Default	0.	0.	0	0	0.	0.		

	1	2	3	4	5	6	7	8
Variable	NODE1	NODE2	D1	D2	D3			
Type	I	I	F	F	F			
Default	0.	0.	0	0	0.			

Optional Card (Required if FINITE is specified after the keyword). See Figure 22.3.

The vector is computed as the vector produce **m**=**n** X **l**. The origin, the taile of the normal vector, is taken as the corner point of the finite size plane.

	1	2	3	4	5	6	7	8
Variable	XHEV	YHEV	ZHEV	LENL	LENM			
Type	F	F	F	F	F			
Default	0.	0.	0.	infinity	infinity			

Optional Card (Required if MOVING is specified after keyword-the MOVING option is not compatible with the ORTHO option).

	1	2	3	4	5	6	7	8
Variable	MASS	V0						
Type	F	F						
Default	none	0.						

Optional Card (Required if FORCES is specified after the keyword)

This option allows the force distribution to be monitored on the plane. Also four points can be defined for visualization of the rigid wall. A shell or membrane element must be defined with these four points as the connectivity for viewing in LS-TAURUS.

	1	2	3	4	5	6	7	8
Variable	SOFT	SSID	N_1	N_2	N_3	N_4		
Type	I	I	I	I	I	I		
Default	0	0	0	0	0	0		
Remarks		1	2					

VARIABLE	DESCRIPTION
NSID	Nodal set ID containing slave nodes, see *SET_NODE_OPTION: EQ.0: all nodes are slave to rigid wall.
NSIDEX	Nodal set ID containing nodes that exempted as slave nodes, see *SET_NODE_OPTION.
BOXID	All nodes in box are included as slave nodes to rigid wall, see *DEFINE_BOX.
XT	x-coordinate of tail of any outward drawn normal vector, **n**, originating on wall (tail) and terminating in space (head), see Figure 22.3.
YT	y-coordinate of tail of normal vector **n**
ZT	z-coordinate of tail of normal vector **n**
XH	x-coordinate of head of normal vector **n**
YH	y-coordinate of head of normal vector **n**
ZH	z-coordinate of head of normal vector **n**
FRIC	Interface friction: EQ.0.0: frictionless sliding after contact, EQ.1.0: no sliding after contact, 0.<FRIC<1.: Coulomb friction coefficient.
SFRICA	Static friction coefficient in local a-direction, μ_{sa}, see Figure 22.2.

VARIABLE	DESCRIPTION
SFRCIB	Static friction coefficient in local b-direction, μ_{sb}
DFRICA	Dynamic friction coefficient in local a-direction, μ_{ka}
DFRICB	Dynamic friction coefficient in local b-direction, μ_{kb}
DECAYA	Decay constant in local a-direction, d_{va}
DECAYB	Decay constant in local b-direction, d_{vb}
NODE1	Node 1, alternative to definition with vector d below, see Figure 22.2. With the node definition the direction changes if the nodal pair rotates.
NODE2	Node 2
D1	d_1, x-component of vector, alternative to definition with nodes above, see Figure 22.2. This vector is fixed as a funtion of time.
D2	d_2, y-component of vector
D3	d_3, z-component of vector
XHEV	x-coordinate of head of edge vector **l**, see Figure 22.3.
YHEV	y-coordinate of head of edge vector **l**
ZHEV	z-coordinate of head of edge vector **l**
LENL	Length of **l** edge
LENM	Length of **m** edge
MASS	Total mass of stonewall
V0	Initial velocity of stonewall in direction of defining vector, **n**
SOFT	Number of cycles to zero relative velocity to reduce force spike
SSID	Segment set identification number for defining areas for force output, see *SET_SEGMENT and remark 1 below.
N1	Optional nodal point for visualization in LS-TAURUS database, see remark 2 below.
N2	Optional nodal point for visualization
N3	Optional nodal point for visualization
N4	Optional nodal point for visualization

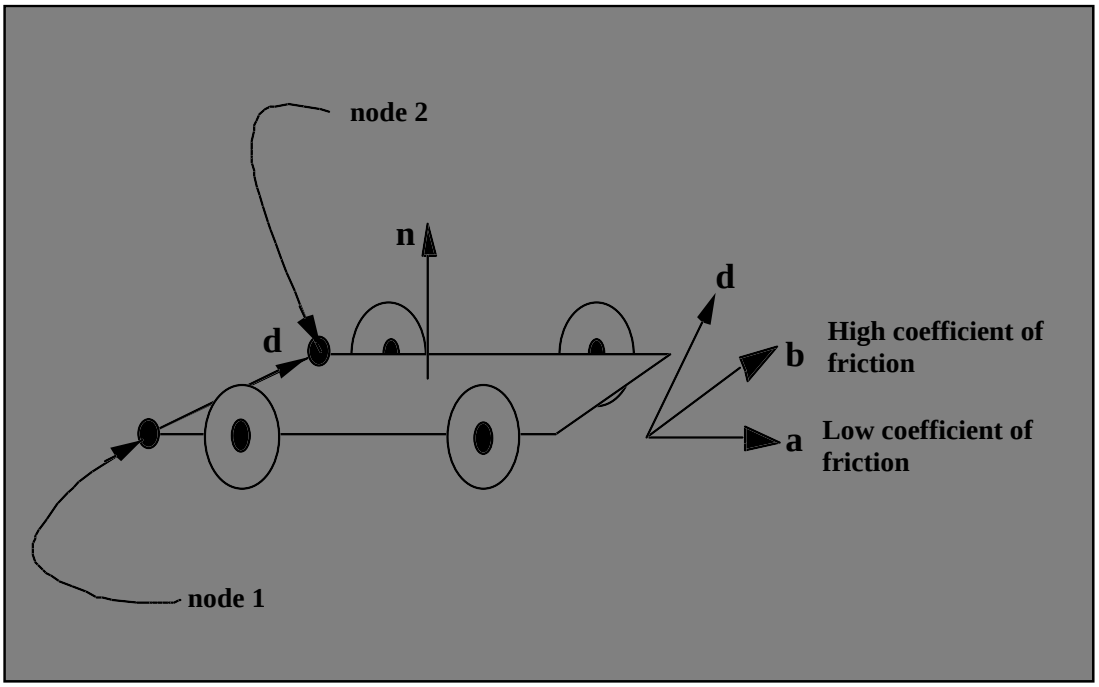


Figure 22.2. Definition of orthotropic friction vectors. The two methods of defining the vector, **d** are shown. If vector **d** is defined by nodes 1 and 2, the local coordinate system may rotate with the body which contains the nodes; otherwise, **d** is fixed in space, thus on the rigid wall, and the local system is stationary.

The coefficients of friction are defined in terms of the static, dynamic and decay coefficients and the relative velocities in the local a and b directions as

$$\mu_a = \mu_{ka} + (\mu_{sa} - \mu_{ka})e^{-d_{va}V_{\text{relative, a}}}$$
$$\mu_b = \mu_{kb} + (\mu_{sb} - \mu_{kb})e^{-d_{vb}V_{\text{relative, b}}}$$

Orthotropic rigid walls can used to model rolling objects on rigid walls where the frictional forces are substantially higher in a direction transverse to the rolling direction. To use this option define a vector **d** to determine the local frictional directions via:

$$\underset{\sim}{b} = \underset{\sim}{n} \times \underset{\sim}{d} \text{ and that } \underset{\sim}{a} = \underset{\sim}{b} \times \underset{\sim}{n}$$

where **n** is the normal vector to the rigid wall. If **d** is in the plane of the rigid wall, the **a** is identical to **d**. See Figure 22.3 below.

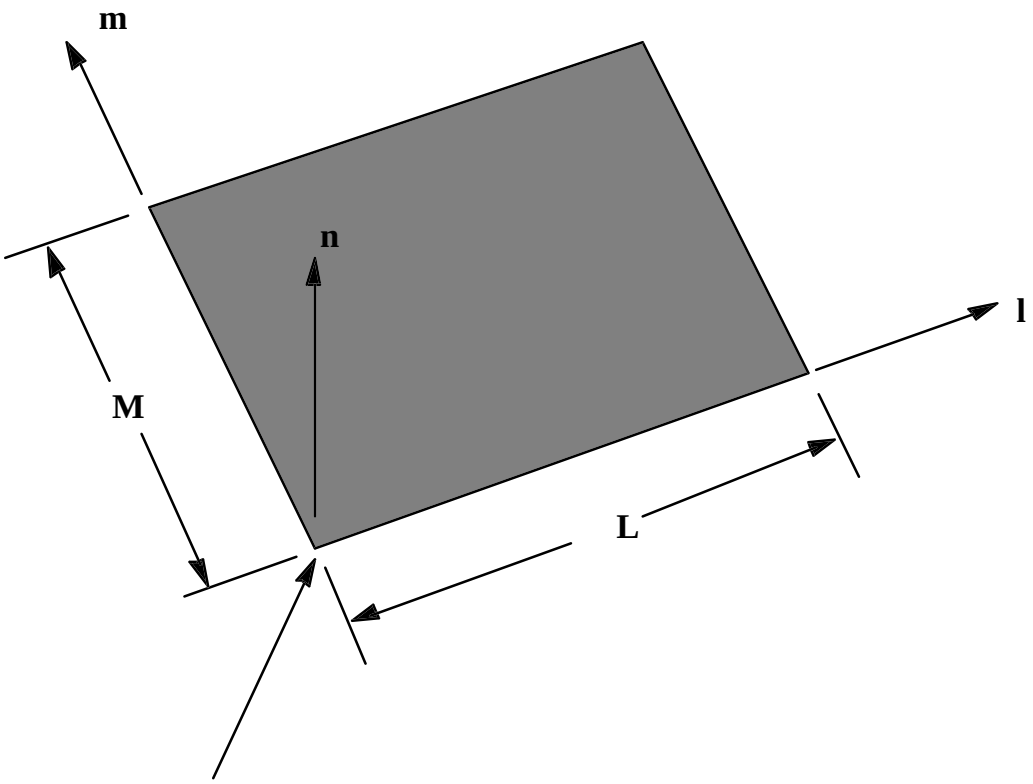


Figure 22.3. Vector **n** is normal to the stonewall. An optional vector **l** can be defined such that **m**=**n** × **l**. The extent of the stonewall is limited by defining **L** (LENL) and **M** (LENM). A zero value for either of these lengths indicates that the stonewall is infinite in that direction.

Remarks:

1 The segment set defines areas for computing resultant forces. These segments translate with the moving stonewall and allow the forced distribution to be determined. The resultant forces are written in file "RWFORC."

2. These four nodes are for visualizing the movement of the wall. They move with the wall. To view the wall in LS-TAURUS it is necessary to define a shell element with these four nodes as its connectivity.

*SECTION

In this section the element formulation, integration rule, nodal thicknesses or cross sectional properties are defined. All section identifiers (SECID's) defined in this section must be unique, i.e., if a number is used as a section ID for a beam then this number cannot be used again even for as a section ID for a solid. The keyword cards in this section are defined in alphabetical order:

*SECTION_BEAM
*SECTION_DISCRETE
*SECTION_SEATBELT
*SECTION_SHELL
*SECTION_SOLID
*SECTION_SOLID_ALE
*SECTION_TSHELL

The location and order of these cards in the input file are arbitrary.

*SECTION

*SECTION_BEAM

Purpose: Define cross sectional properties for beam, truss, discrete beam, and cable elements.

Card Format

Card 1

Variable	SECID	ELFORM	SHRF	QR/IRID	CST			
Type	I	I	F	F	F			
Default	none	1	1.0	2.0	0.0			

Card 2

Integrated	TS1	TS2	TT1	TT2	NSLOC	NTLOC		
Resultant	A	ISS	ITT	IRR	SA			
Discrete	VOL	INER	CID	CA	OFFSET			
Type	F	F	F	F	F	F		

VARIABLE	DESCRIPTION
SECID	Section ID. SECID is referenced on the *PART card and must be unique.
ELFORM	Element formulation options: EQ.1: Hughes-Liu with cross section integration (default), EQ.2: Belytschko-Schwer resultant beam (resultant), EQ.3: truss (resultant), EQ.4: Belytschko-Schwer full cross-section integration, EQ.5: Belytschko-Schwer tubular beam with cross-section integration, EQ.6: discrete beam/cable.
SHRF	Shear factor. This factor is not needed for truss, resultant beam, discrete beam, and cable elements. The recommended value for rectangular sections is 5/6, the default is 1.0.

VARIABLE	DESCRIPTION
QR/IRID	Quadrature rule or rule number for user defined rule for integrated beams: EQ.1.0: one integration point, EQ.2.0: 2×2 Gauss quadrature (default beam), EQ.3.0: 3×3 Gauss quadrature, EQ.4.0: 3×3 Lobatto quadrature, EQ.5.0: 4×4 Gauss quadrature EQ.-n: where |n| is the number of the user defined rule. IRID integration rule n is defined using *INTEGRATION_BEAM card.
CST	Cross section type, not needed for truss, resultant beam, discrete beam, and cable elements: EQ.0.0: rectangular, EQ.1.0: tubular, EQ.2.0: arbitrary (user defined integration rule).
TS1	Beam thickness (CST=0.0, 2.0) or outer diameter (CST = 1.0) in s direction at node n_1. Note that the thickness defined on the *BEAM_ ELEMENT_THICKNESS card overrides the definition give here.
TS2	Beam thickness (CST=0.0, 2.0) or outer diameter (CST = 1.0) in s direction at node n_2.
TT1	Beam thickness (CST=0.0, 2.0) or inner diameter (CST = 1.0) in t direction at node n_1.
TT2	Beam thickness (CST=0.0, 2.0) or inner diameter (CST = 1.0) in t direction at node n_2.
NSLOC	Location of reference surface normal to s axis for Hughes-Liu beam elements only: EQ.1.0: side at s=1.0, EQ.0.0: center, EQ.-1.0: side at s = -1.0.
NTLOC	Location of reference surface normal to t axis for Hughes-Liu beam elements only: EQ.1.0: side at t =1.0, EQ.0.0: center, EQ.-1.0: side at t = -1.0.
A	Cross-sectional area. The definition on *BEAM_ELEMENT_THICKNESS overrides the value defined here, see Figure 23.1.
ISS	I_{ss}. The definition on *BEAM_ELEMENT_THICKNESS overrides the value defined here, see Figure 23.1.

VARIABLE	DESCRIPTION
ITT	I_{tt}. The definition on *BEAM_ELEMENT_THICKNESS overrides the value defined here, see Figure 23.1.
IRR	I_{rr} (J) polar inertia. The definition on *BEAM_ELEMENT_THICKNESS overrides the value defined here, see Figure 23.1.
SA	Shear area. The definition on *BEAM_ELEMENT_THICKNESS overrides the value defined here, see Figure 23.1.
VOL	Volume of discrete beam.
INER	I, lumped inertia of discrete beam which have six degrees of freedom. This lumped inertia is partitioned to the two nodes of the beam element.
CID	Coordinate system ID for orientation, materials type ID (67-69), see *COORDINATE_SYSTEM. This is not defined for cable elements.
CA	Cable area, materials type ID 71, *MAT_CABLE.
OFFSET	Offset for cable. For a definition see materials type ID 71, *MAT_CABLE.

Remark:

1. For the truss element, define the cross-sectional area, A, only.

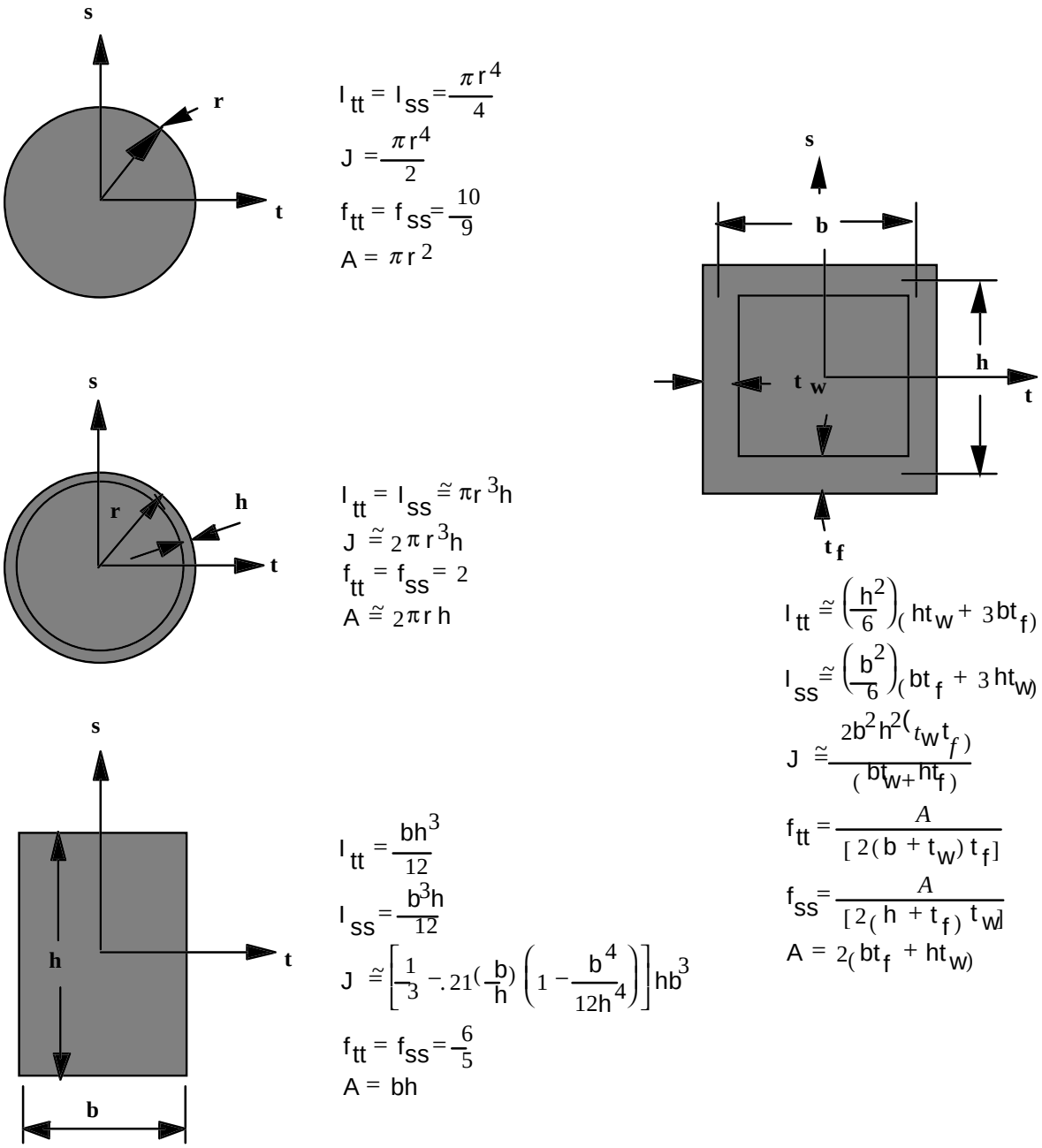


Figure 23.1. Properties of beam cross section for several common cross sections.

*SECTION

*SECTION_DISCRETE

Purpose: Defined spring and damper elements for translation and rotations. See also explanation below. The definitions below have to correspond with the material type selection for the elements.

Card Format

Card 1	1	2	3	4	5	6	7	8
Variable	SECID	DRO	KD	V0	CL	FD		
Type	I	I	F	F	F	F		

Card 2	1	2	3	4	5	6	7	8
Variable	CDL	TDL						
Type	F	F						

VARIABLE	DESCRIPTION
SECID	Section ID. SECID is referenced on the *PART card and must be unique.
DRO	Displacement/Rotation Option: EQ.0: the material describes a translational spring/damper, EQ.1: the material describes a torsional spring/damper.
KD	Dynamic magnification factor
V0	Test velocity
CL	Clearance
FD	Failure deflection (twist for DRO=1).
CDL	Deflection (twist for DRO=1) limit in compression, see comment below.
TDL	Deflection (twist for DRO=1) limit in tension, see comment below.

The constants from KD to TDL are optional and do not need to be defined.

If k_d is nonzero, the forces computed from the spring elements are assumed to be the static values and are scaled by an amplification factor to obtain the dynamic value:

$$F_{dynamic} = \left(1.+k_d \frac{V}{V_0}\right) F_{static}$$

where

V = absolute value of the relative velocity between the nodes.
V_0 = dynamic test velocity.

For example, if it is known that a component shows a dynamic crush force at 15m/s equal to 2.5 times the static crush force, use k_d =1.5 and V_0=15.

Here, "clearance" defines a compressive displacement which the spring sustains before beginning the force-displacement relation given by the load curve defined in the material selection. If a non-zero clearance is defined, the spring is compressive only.

The deflection limit in compression and tension is restricted in its application to no more than one spring per node subject to this limit, and to deformable bodies only. For example in the former case, if three springs are in series, either the center spring or the two end springs may be subject to a limit, but not all three. When the limiting deflection is reached, momentum conservation calculations are performed and a common acceleration is computed in the appropriate direction. An error termination will occur if a rigid body node is used in a spring definition where deflection is limited.

*SECTION

*SECTION_SEATBELT

Purpose: Define section properties for the seat belt elements. This card is required for the *PART Section. Currently nothing but the ID is required.

Card Format

Card 1	1	2	3	4	5	6	7	8
Variable	SECID							
Type	I							

VARIABLE	DESCRIPTION
SECID	Section ID

***SECTION_SHELL**

Purpose: Define section properties for shell elements.

Card Format

Card 1	1	2	3	4	5	6	7	8
Variable	SECID	ELFORM	SHRF	NIP	PROPT	QR/IRID	ICOMP	
Type	I	I	F	F	F	F	I	
Default	none	2	1.0	2	0.0	0.0	0	

Card 2	1	2	3	4	5	6	7	8
Variable	T1	T2	T3	T4	NLOC			
Type	F	F	F	F	F			
Default	0.0	0.0	0.0	0.0	0.0			

Optional Section Cards if ICOMP=1. Define NIP angles putting 8 on each card.

Cards 3,4,..	1	2	3	4	5	6	7	8
Variable	B1	B2	B3	B4	B5	B6	B7	B8
Type	F	F	F	F	F	F	F	F

VARIABLE	DESCRIPTION
SECID	Section ID. SECID is referenced on the *PART card and must be unique.

VARIABLE	DESCRIPTION
ELFORM	Element formulation options: EQ.1: Hughes-Liu, EQ.2: Belytschko-Tsay (default), EQ.3: BCIZ triangular shell, EQ.4: C_0 triangular shell, EQ.5: Belytschko-Tsay membrane, EQ.6: S/R Hughes-Liu , EQ.7: S/R co-rotational Hughes-Liu, EQ.8: Belytschko-Leviathan shell , EQ.9: fully integrated Belytschko-Tsay membrane, EQ.10: Belytschko-Wong-Chiang, EQ.11: Fast (co-rotational) Hughes-Liu.
SHRF	Shear factor. A suggested value is 5/6.
NIP	Number of through shell thickness integration points: EQ.0: set to 2 integration points.
PROPT	Printout option: EQ.1.0: average resultants and fiber lengths, EQ.2.0: resultants at plan points and fiber lengths, EQ.3.0: resultants, stresses at all points, fiber lengths.
QR/IRID	Quadrature rule or Integration rule ID, see *INTEGRATION_SHELL: LT.0.0: absolute value is specified rule number, EQ.0.0: Gauss (up to five points are permitted), EQ.1.0: trapezoidal, *not recommend for accuracy reasons*.
ICOMP	Flag for layered composite material model: EQ.1: a material angle is defined for each through thickness integration point. Thus, each layer has one integration point.
T1	Shell thickness at node n_1, see Figure 10.14. The thickness is defined on the *ELEMENT_SHELL_OPTION card.
T2	Shell thickness at node n_2, see comment for T1 above.
T3	Shell thickness at node n_3, see comment for T1 above.
T4	Shell thickness at node n_4, see comment for T1 above.
NLOC	Location of reference surface (Hughes-Liu shell only): EQ.1.0: top surface, EQ.0.0: mid surface (default), EQ.-1.0: bottom surface.
B1	β_1, material angle at first integration point
B2	β_2, material angle at second integration point

VARIABLE	DESCRIPTION
B3	β_3, material angle at third integration point
.	.
.	.
.	.
B8	β_8, material angle at eigth integration point
.	.
Bnip	β_{nip}, material angle at nipth integration point

*SECTION

***SECTION_SOLID_OPTION**

Options include:

<BLANK>
ALE

such that the keyword cards appear:

***SECTION_SOLID**
***SECTION_SOLID_ALE**

Purpose: Define section properties for solid continuum and fluid elements.

Card 1 define for all options

Card 1	1	2	3	4	5	6	7	8
Variable	SECID	ELFORM	AET					
Type	I	I	I					

Card 2 define only for the ALE option.
Also see *ALE_SMOOTHING for the smoothing definition.

Cards 2	1	2	3	4	5	6	7	8
Variable	AFAC	BFAC	CFAC	DFAC	START	END	AAFAC	
Type	F	F	F	F	F	F	F	

VARIABLE	DESCRIPTION
SECID	Section ID. SECID is referenced on the *PART card and must be unique.
ELFORM	Element formulation options: EQ.1: constant stress solid element (default), EQ.2: fully integrated S/R solid, EQ.3: fully integrated quadratic eight node element with nodal rotations, EQ.4: S/R quadratic tetrahedron element with nodal rotations, EQ.5: 1 point ALE, EQ.6: 1 point Eulerian, EQ.7: 1 point Eulerian ambient, EQ.8: acoustic.
AET	Ambient Element type: EQ.1: temperature, EQ.2: pressure and temperature, EQ.3: pressure outflow, EQ.4: pressure inflow (default).
AFAC	Smoothing weight factor - Simple average: EQ.-1: turn smoothing off.
BFAC	Smoothing weight factor - Volume weighting
CFAC	Smoothing weight factor - Isoparametric
DFAC	Smoothing weight factor - Equipotential
START	Start time for smoothing
END	End time for smoothing
AAFAC	ALE advection factor

*SECTION

*SECTION_TSHELL

Purpose: Define section properties for thick shell elements.

Card Format

Card 1	1	2	3	4	5	6	7	8
Variable	SECID	ELFORM	SHRF	NIP	PROPT	QR	ICOMP	
Type	I	I	F	F	F	F	I	
Default	none	1	1.0	2	1	0	0	

Optional Section Cards if ICOMP=1 define NIP angles putting 8 on each card.

Cards 2,3,..	1	2	3	4	5	6	7	8
Variable	B1	B2	B3	B4	B5	B6	B7	B8
Type	F	F	F	F	F	F	F	F

VARIABLE	DESCRIPTION
SECID	Section ID. SECID is referenced on the *PART card and must be unique.
ELFORM	Element formulation: EQ.1: one point reduced integration (default), EQ.2: selective reduced 2×2 in plane integration.
SHRF	Shear factor. A value of 5/6 is recommended.
NIP	Number of through shell thickness integration points: EQ.0: set to 2 integration points.
PROPT	Printout option: EQ.1.0: average resultants and fiber lengths, EQ.2.0: resultants at plan points and fiber lengths, EQ.3.0: resultants, stresses at all points, fiber lengths.

VARIABLE	DESCRIPTION
QR	Quadrature rule: LT.0.0: absolute value is specified rule number, EQ.0.0: Gauss (up to five points are permitted), EQ.1.0: trapezoidal, not recommended for accuracy reasons.
ICOMP	Flag for layered composite material mode: EQ.1: a material angle is defined for each through thickness integration point . For each layer one integration point is used.
B1	β_1, material angle at first integration point. The same procedure for determining material directions is use for thick shells that is used for the 4 node quadrilateral shell.
B2	β_2, material angle at second integration point
B3	β_3, material angle at third integration point
.	.
.	.
.	.
B8	β_8, material angle at eigth integration point
.	.
Bnip	β_{nip}, material angle at nipth integration point

Define as many cards as necessary until NIP points are defined.

*SET

The keyword *SET provides a convenient way of defining groups of nodes, parts, elements, and segments. The sets can be used in the definitions of contact interfaces, loading conditions, boundary condtions, and other inputs. Each set type is numbered separately. The keyword control cards in this section are defined in alphabetical order:

*SET_BEAM
*SET_DISCRETE
*SET_NODE_OPTION
*SET_PART_OPTION
*SET_SEGMENT
*SET_SHELL_OPTION
*SET_SOLID
*SET_TSHELL

***SET_BEAM**

Purpose: Define a set of beam elements.

Card Format

Card 1	1	2	3	4	5	6	7	8
Variable	SID							
Type	I							
Default	none							

Card 2, 3, 4, ... (The next "*" card terminates the input.)

	1	2	3	4	5	6	7	8
Variable	K1	K2	K3	K4	K5	K6	K7	K8
Type	I	I	I	I	I	I	I	I

VARIABLE	DESCRIPTION
SID	Set ID
K1	First beam element
K2	Second beam element
.	. .
.	. .
KNUM	Last beam element

***SET_DISCRETE**

Purpose: Define a set of discrete elements.

Card Format

Card 1	1	2	3	4	5	6	7	8
Variable	SID							
Type	I							
Default	none							

Card 2, 3, 4, ... (The next "*" card terminates the input.)

	1	2	3	4	5	6	7	8
Variable	K1	K2	K3	K4	K5	K6	K7	K8
Type	I	I	I	I	I	I	I	I

VARIABLE	DESCRIPTION
SID	Set ID
K1	First discrete element
K2	Second discrete element
.	. .
.	. .
KNUM	Last discrete element

***SET_NODE_OPTION**

Available options include:

LIST
COLUMN

Purpose: Define a nodal set with some identical or unique attributes.

Card Format

	1	2	3	4	5	6	7	8
Variable	SID	DA1	DA2	DA3	DA4			
Type	I	F	F	F	F			
Default	none	0.	0.	0.	0.			
Remark		1	1	1	1			

Cards 2, 3, 4, ... (OPTION=LIST) (The next "*" card terminates the input.)

	1	2	3	4	5	6	7	8
Variable	NID1	NID2	NID3	NID4	NID5	NID6	NID7	NID8
Type	I	I	I	I	I	I	I	I

Cards 2, 3, 4, ... (OPTION=COLUMN) (The next "*" card terminates the input.)

	1	2	3	4	5	6	7	8
Variable	NID	A1	A2	A3	A4			
Type	I	F	F	F	F			
Remark		2	2	2	2			

VARIABLE	DESCRIPTION
SID	Set identification. All node sets should have a unique set ID.
DA1	First nodal attribute default value, see remark 1 below.
DA2	Second nodal attribute default value
DA3	Third nodal attribute default value
DA4	Fourth nodal attribute default value
NIDN	Node ID n
NID	Nodal ID
A1	First nodal attribute, see remark 2 below.
A2	Second nodal attribute
A3	Third nodal attribute
A4	Fourth nodal attribute

Remarks:

1. Nodal attributes can be assigned for some input types. For example, for contact option, *CONTACT_TIEBREAK_NODES_TO_SURFACE the attributes are:
   - DA1=NFLF Normal failure force,
   - DA2=NSFLF Shear failure force,
   - DA3=NNEN Exponent for normal force,
   - DA4=NMES Exponent for shear force.
2. The default nodal attributes can be overridden on these cards; otherwise, A1=DA1, etc.

***SET_PART_OPTION**

Available options include:

LIST
COLUMN

Purpose: Define a set of parts with optional attributes. For the column option, see *AIRBAG or *CONSTRAINED _RIGID_BODY_STOPPERS.

Card Format

	1	2	3	4	5	6	7	8
Variable	SID	DA1	DA2	DA3	DA4			
Type	I	F	F	F	F			
Default	none	0.						
Remark		1	1	1	1			

Card 2, 3, 4, ... (OPTION=LIST) (The next "*" card terminates the input.)

	1	2	3	4	5	6	7	8
Variable	PID1	PID2	PID3	PID4	PID5	PID6	PID7	PID8
Type	I	I	I	I	I	I	I	I

Card 2, 3, 4, ... (OPTION=COLUMN) (The next "*" card terminates the input.)

	1	2	3	4	5	6	7	8
Variable	PID	A1	A2	A3	A4			
Type	I	F	F	F	F			
Remark		1	1	1	1			

VARIABLE	DESCRIPTION
SID	Set ID. All part sets should have a unique set ID.
DA1	First attribute default value, see remark 1 below.
DA2	Second attribute default value
DA3	Third attribute default value
DA4	Fourth attribute default value
PID	Part ID
PID1	First part ID
PID2	Second part ID
.	. .
A1	First part attribute, see remark 2 below.
A2	Second part attribute
A3	Third part attribute
A4	Fourth part attribute

Remarks:

1. Part attributes can be assigned for some input types. For example, for airbags a time delay, DA1=T1, can be defined before pressure begins to act along with a time delay, DA2=T2, before full pressure is applied, (default T2=T1), and for the constraint option, *CONSTRAINED_RIGID_

BODY_STOPPERS one attribute can be defined: DA1, the closure distance which activates the stopper constraint.

2. The default part attributes can be overridden on the part cards; otherwise, A1=DA1, etc.

***SET_SEGMENT**

Purpose: Define a set of quadrilateral and triangular segments with optional identical or unique attributes.

Card Format

	1	2	3	4	5	6	7	8
Variable	SID	DA1	DA2	DA3	DA4			
Type	I	F	F	F	F			
Default	none	0.	0.	0.	0.			
Remarks		1	1	1	1			

Cards 2, 3, 4, ... (The next "*" card terminates the input.)

	1	2	3	4	5	6	7	8
Variable	N1	N2	N3	N4	A1	A2	A3	A4
Type	I	I	I	I	F	F	F	F
Remarks				2	3	3	3	3

VARIABLE	DESCRIPTION
SID	Set ID. All segment sets should have a unique set ID.
DA1	First segment attribute default value, see remark 1 below.
DA2	Second segment attribute default value
DA3	Third segment attribute default value
DA4	Fourth segment attribute default value

VARIABLE	DESCRIPTION
N1	Nodal point n_1
N2	Nodal point n_2
N3	Nodal point n_3
N4	Nodal point n_4, see remark 2 below.
A1	First segment attribute, see remark 3 below.
A2	Second segment attribute
A3	Third segment attribute
A4	Fourth segment attribute
NFLS	Normal failure stress
SFLS	Shear failure stress. Failure criterion:

Remarks:

1. Segment attributes can be assigned for some input types. For example, for the contact options, the attributes for the SLAVE surface are:
   - DA1=NFLS Normal failure stress, *CONTACT_TIEBREAK_SURFACE_contact only,
   - DA2=SFLS Shear failure stress, *CONTACT_TIEBREAK_SURFACE_contact only,
   - DA3=FSF Coulomb friction scale factor,
   - DA4=VSF Viscous friction scale factor,

   and the attributes for the MASTER surface are:
   - DA1=FSF Coulomb friction scale factor,
   - DA2=VSF Viscous friction scale factor.

   For airbags, see *AIRBAG, a time delay, DA1=T1, can be defined before pressure begins to act on a segment along with a time delay, DA2=T2, before full pressure is applied to the segment, (default T2=T1), and for the constraint option,

2. To define a triangular segment make n_4 equal to n_3.

3. The default segment attributes can be overridden on these cards, otherwise, A1=DA1, etc.

***SET_SHELL_OPTION**

Available options include:

LIST
COLUMN

Purpose: Define a set of shell elements with optional identical or unique attributes.

Card Format

	1	2	3	4	5	6	7	8
Variable	SID	DA1	DA2	DA3	DA4			
Type	I	F	F	F	F			
Default	none	0.	0.	0.	0.			
Remarks		1	1	1	1			

Card 2, 3, 4, ... (OPTION=LIST) (The next "*" card terminates the input.)

	1	2	3	4	5	6	7	8
Variable	EID1	EID2	EID3	EID4	EID5	EID6	EID7	EID8
Type	I	I	I	I	I	I	I	I
Remarks	2	2	2	2	2	2	2	2

Card 2, 3, 4, ... (OPTION=COLUMN) (The next "*" card terminates the input.)

	1	2	3	4	5	6	7	8
Variable	EID	A1	A2	A3	A4			
Type	I	F	F	F	F			
Remarks		3	3	3	3			

VARIABLE	DESCRIPTION
SID	Set ID. All shell sets should have a unique set ID.
DA1	First attribute default value, see remark 1.
DA2	Second attribute default value
DA3	Third attribute default value
DA4	Fourth attribute default value
EID1	First shell element ID, see remark 2.
EID2	Second shell element ID
.	. .
.	. .
EID	Element ID
A1	First attribute
A2	Second attribute
A3	Third attribute
A4	Fourth attribute

Remarks:

1. Shell attributes can be assigned for some input types. For example, for the contact options, the attributes for the SLAVE surface are:

   DA1=NFLS Normal failure stress, *CONTACT_TIEBREAK_SURFACE_contact only,

   DA2=SFLS Shear failure stress, *CONTACT_TIEBREAK_SURFACE_contact only,

   DA3=FSF Coulomb friction scale factor,

   DA4=VSF Viscous friction scale factor,

   and the attributes for the MASTER surface are:

   DA1=FSF Coulomb friction scale factor,

   DA2=VSF Viscous friction scale factor.

2. The default attributes are taken.

3. The default shell attributes can be overridden on these cards; otherwise, A1=DA1, etc.

***SET_SOLID**

Purpose: Define a set of solid elements.

Card Format

	1	2	3	4	5	6	7	8
Variable	SID							
Type	I							
Default	none							

Card 2, 3, 4, ...(The next "*" card terminates the input.)

	1	2	3	4	5	6	7	8
Variable	K1	K2	K3	K4	K5	K6	K7	K8
Type	I	I	I	I	I	I	I	I

VARIABLE	DESCRIPTION
SID	Set ID. All solid sets should have a unique set ID.
K1	First element ID
K2	Second element ID
.	. .
.	. .
K8	Eighth element ID
.	. .
.	. .

***SET_TSHELL**

Purpose: Define a set of thick shell elements.

Card Format

	1	2	3	4	5	6	7	8
Variable	SID							
Type	I							
Default	none							

Card 2, 3, 4, ... (The next "*" card terminates the input.)

	1	2	3	4	5	6	7	8
Variable	K1	K2	K3	K4	K5	K6	K7	K8
Type	I	I	I	I	I	I	I	I

VARIABLE	DESCRIPTION
SID	Set ID. All tshell sets should have a unique set ID.
K1	First thick shell element ID
K2	Second thick shell element ID
. . .	. . .
K8	Eighth thick shell element ID
. . .	. . .

*TERMINATION

***TERMINATION_OPTION**

Available options include:

NODE
BODY

Caution: The inputs are different for the nodal and rigid body stop conditions. The nodal stop condition works on the global coordinate position, while the body stop condition works on the relative global translation.

The analysis terminates for *TERMINATION_NODE when the current position of the node specified reaches either the maximum or minimum value (stops 1, 2 or 3), or picks up force from any contact surface (stop 4). For *TERMINATION_BODY the analysis terminates when the centre of mass displacement of the rigid body specified reaches either the maximum or minimum value (stops 1, 2 or 3) or the displacement magnitude of the centre of mass is exceeded (stop 4). If more than one condition is input, the analysis stops when any of the conditions is satisfied. Termination by other means is controlled by the *CONTROL_TERMINATION control card.

Note: This type of termination is not active during dynamic relaxation.

*TERMINATION

Card Format

	1	2	3	4	5	6	7	8
Variable	NID/PID	STOP	MAXC	MINC				
Type	I	I	F	F				
Default	none	none	-	-				

For the **NODE** option:

VARIABLE	DESCRIPTION
NID	Node ID, see *NODE_OPTION.
STOP	Stop criterion: EQ.1: global x direction, EQ.2: global y direction, EQ.3: global z direction, EQ.4: stop if node touches contact surface.
MAXC	Maximum (most positive) coordinate (options 1, 2 and 3) above only.
MINC	Minimum (most negative) coordinate (options 1, 2 and 3) above only.

For the **BODY** option:

VARIABLE	DESCRIPTION
PID	Part ID of rigid body, see *PART_OPTION.
STOP	Stop criterion: EQ.1: global x direction, EQ.2: global y direction, EQ.3: global z direction, EQ.4: stop if displacement magnitude is exceeded.
MAXC	Maximum (most positive) displacement, options 1, 2, 3 and 4: EQ:0.0. MAXC set to 1.0e21.
MINC	Minimum (most negative) displacement, options 1, 2 and 3 above only: EQ:0.0. MINC set to -1.0e21.

*TITLE

***TITLE**

Purpose: Define job title.

Card Format

1	2	3	4	5	6	7	8

Variable	TITLE
Type	C
Default	LS-DYNA3D USER INPUT

VARIABLE	DESCRIPTION
TITLE	Heading to appear on output and in output files.

*TRANSLATE

***TRANSLATE_ANSYS_OPTION**

Available options include:

4

5

corresponding to ANSYS version numbers 4 and 5.

Purpose: Provide a convenient route to read in ANSYS input decks as part of the LS-DYNA3D keyword input. This keyword can appear more than once anywhere in the input. It is a direct interface to ANSYS file28 keyword files.

Card Format

	1	2	3	4	5	6	7	8
Variable	FILE							
Type	A							
Default	none							

VARIABLE	DESCRIPTION
FILE	Filename of file created by ANSYS, see remarks below.

The supported options include:

Version	ANSYS Keyword	LS-DYNA3D Keyword
All	N,Type,NODE,Val1,Val2,Val3	*NODE
All	EN,Type,I1,I2,I3,I4,I5,I6,I7,I8	*ELEMENT
All	MPDATA, R5.0, LENGTH, Lab, MAT, STLOC, VAL1, VAL2, VAL3	*MAT_ELASTIC

*TRANSLATE

Version	ANSYS Keyword	LS-DYNA3D Keyword
All	ET, Type	*PART&*SECTION
All	R,R5.0,NSET,Type,STLOC,VAL1,VAL2,VAL3	*PART&*SECTION
5	DFLAB,NODF,LabD,LabF	
5	NDOF.eq.Ui,ROTi; LabD.eq.0	*BOUNDARY_SPC_option
5	NODF.eq.Vi; LabD.eq.0	*INITIAL_VELOCITY_NODE
5	NODF.eq.Ui,ROTi,Ai,Vi,;LabD.eq.lcid; LabF.eq.val	*BOUNDARY_PRESCRIBED_MOTION_NODE
5	NDOF.eq.Fi; LabF.eq.lcid	*LOAD_NODE_POINT
5	SFE,ELEM,LKEY,Lab,KEY,R5.0	
5	LKEY.eq.lcid; Lab.eq.pressure	*LOAD_SEGMENT

Remarks:

1. Supported keywords as described in the SASI ANSYS Manual chapter on "Exporting a Finite Element Model."

2. Solid elements and shell elements cannot have the same R value in reference to the ET and R ANSYS keywords.

3. Supported element types include: 63.eq.shells, 45.eq.solids, 73.eq.solids, 4.eq.beams, 16.eq.pipes, and 21.eq.lumped masses.

***TRANSLATE_IDEAS_OPTION**

Available options include:

MASTER

Purpose: Provide a convenient route to read in files created by IDEAS/SUPERTAB as part of the LS-DYNA3D keyword input. This keyword can appear more than once in the input. It is a direct interface to IDEAS universal files.

Card Format

	1	2	3	4	5	6	7	8
Variable	FILE							
Type	A							
Default	none							

VARIABLE	DESCRIPTION
FILE	Filename of the IDEAS universal file.

The following table lists supported IDEAS keywords:

Version	SDRC IDEAS Universal File	LS-DYNA3D Keyword
All	N,Type,NODE,Val1,Val2,Val3	*NODE
All	EN,Type,I1,I2,I3,I4,I5,I6,I7,I8	*ELEMENT
5	781	*NODE
7	2411	*NODE
5	780	*ELEMENT
7	2412	*ELEMENT
5	773	*MAT_ELASTIC
5	772	*PART&*SECTION
6	788	*PART&*SECTION

*TRANSLATE

Version	SDRC IDEAS Universal File	LS-DYNA3D Keyword
7	2430	*PART&*SECTION
5	755	*BOUNDARY_SPC_NODE
7	791	
	time variation set.le.0.0	*BOUNDARY_SPC_NODE
	time variation set.gt.0.0	*BOUNDARY_PRESCRIBED_ MOTION_NODE
7	790	
	load type.eq.1	*LOAD_NODE

***TRANSLATE_NASTRAN**

Purpose: Provide a convenient route to read in NASTRAN input deck as part of the LSDYNA3D keyword input. This keyword can appear more than once anywhere in the input. Also, see remarks below.

Card Format

	1	2	3	4	5	6	7	8
Variable	FILE							
Type	C							

VARIABLE	DESCRIPTION
FILE	Filename of the NASTRAN input deck.

The following table lists supported NASTRAN keywords:

Version	NASTRAN INPUT FILE	LS-DYNA3D Keyword
All	N,Type,NODE,Val1,Val2,Val3	*NODE
All	EN,Type,I1,I2,I3,I4,I5,I6,I7,I8	*ELEMENT
All	BEGIN BULK	
All	GRID	*NODE
All	CORD2R	*DEFINE_COORDINATE_SYSTEM
All	CHEXA, CPENTA, CTETRA	*ELEMENT_SOLID
All	PSOLID	*PART and *SECTION_SOLID
All	CQUAD4, CTRIA3	*ELEMENT_SHELL
All	PSHELL	*PART and *SECTION_SHELL
All	CBAR, CBEAM	*ELEMENT_BEAM
All	CELAS1, CVISC, CDAMP1	*ELEMENT_DISCRETE
All	CONM2	*ELEMENT_MASS
All	MAT1	*MAT_ELASTIC
All	SPC, SPC1	*BOUNDARY_SPC_OPTIONS

*TRANSLATE

Version	NASTRAN INPUT FILE	LS-DYNA3D Keyword
All	RBE2	*CONSTRAINED_NODE_SET or *CONSTRAINED_NODAL_RIGID_BODY_
All	ENDDATA	*END

Remarks:

1. Both small and large field fixed NASTRAN formats are supported.

2. The same keywords in LS-DYNA3D usually contain more options than the NASTRAN input. Therefore, to make it complete, we add some extra parameters to the NASTRAN keywords. For those extras we use the italics to distinguish from the standard ones. These additional parameters have to be added to the NASTRAN deck by the user to make the translation complete.

Card Format

For further explanation see *ELEMENT_DISCRETE.

	1	2	3	4	5	6	7	8	9
CELAS1	EID	PID	*N1*	*DFG*	*N2*	*VID*	*S*	*PF*	
Type	I	I	I	F	I	F	F	F	

For further explanation see *PART and *SECTION_SHELL.

PSHELL	PID	MID	*T*	*FORM*	*SHEAR*	*NIP*	*QR*	*HGID*	
Type	I	I	F	I	F	I	F	I	

3. Current **NASTRAN** only supports shell element with constant thickness T.

For further explanation see *PART and *SECTION_SOLID.

PSOLID	PID	MID	*SCID*	*EOSID*	*HGID*				
Type	I	I	I	I	I				

4. The **THRU** command for SPC, SPC1 is not supported in the current translation.

5. For RBE2 keyword, if any of the rotational DOF (4,5,6) appears in the constraint, LS-DYNA3D will treat it as nodal rigid body constraint. Otherwise, LS-DYNA3D will use nodal constraints to treat this RBE2.

*USER

***USER_INTERFACE_OPTION**

Options include:

CONTROL
FRICTION

Purpose: Define user defined input and allocate storage for user defined subroutines for the contact algorithms. See also *CONTROL_CONTACT. The **CONTROL** option above allows the user to take information from the contact interface for further action, e.g., stopping the analysis. A sample user subroutine is provided in Appendix D.

The **FRICTION** option may be used to modify the Coulomb friction coefficients according to contact information or to use a friction coefficient database. A sample subroutine for treating the friction in contact is provided in Appendix E.

Card Format

	1	2	3	4	5	6	7	8
Variable	IFID	NOC	NOCI					
Type	I	I	I					
Default	none	none	none					

Card Format (Use as many cards as necessary to define NOCI variables)

	1	2	3	4	5	6	7	8
Variable	UC1	UC2	UC3	UC4	UC5	UC6	UC7	UC8
Type	F	F	F	F	F	F	F	F
Default	0.	0.	0.	0.	0.	0.	0.	0.

VARIABLE	DESCRIPTION
IFID	Interface number
NOC	Number of history variables for interface. The number should not exceed the length of the array defined on *CONTROL_CONTACT.
NOCI	Initialize the first NOCI history variables in the input. NOCI must be smaller or equal to NOC.
UC1	First user defined input parameter
UC2	Second user defined input parameter
.	. .
.	. .
.	. .
UCNOCI	Last user defined input parameter
.	. .
.	. .
.	. .

***USER_LOADING**

Purpose: Provide a means of applying pressure and force boundary conditions. The keyword *USER_LOADING activates this option. Input here is optional with the input being read until the next "*" keyword appears. The data read here is to be stored in a common block provided in the user subroutine. This data is stored and retrieved from the restart files.

Card Format (Insert as many cards as needed. The next * card terminates input.)

	1	2	3	4	5	6	7	8
Variable	PARM1	PARM2	PARM3	PARM4	PARM5	PARM6	PARM7	PARM8
Type	F	F	F	F	F	F	F	F
Default	none	none	none	none	none	none	none	none

VARIABLE	DESCRIPTION
PARMn	This is the nth user input parmeter.

RESTART INPUT DATA

In general three categories of restart actions are possible with LS-DYNA3D and are outlined in the following discussion:

a) A simple restart occurs when LS-DYNA3D was interactively stopped before reaching the termination time. Then simply defining the R=rtf file on the execution line for LS-DYNA3D restarts the calculation from the termnination point and the calculation will continue to the specified termination time-see INTRODUCTION, Execution Syntax. No additional input deck is required.

b) If minor modifications are desired as, e.g.,

- reset termination time,
- reset output printing interval,
- reset output plotting interval,
- delete contact surfaces,
- delete elements and parts,
- switch deformable bodies to rigid,
- switch rigid bodies to deformable,
- change damping options.

This type of restart is called a small restart and the corresponding input deck a "small restart input deck." All modifications to the problem made with the restart input deck will be reflected in subsequent restart dumps. All the members of the file families are consecutively numbered beginning from the last member. The small input deck replaces the standard input deck on the execution line which has at least the following contents:

LS-DYNA3D I=*restartinput* R=*D3DUMPnn*

where *D3DUMPnn* (or whatever name is chosen for the family member) is the *n* th restart file from the last run where the data is taken. LS-DYNA3D automatically detects that a small input deck is used since the I=*restartinput* file may contain the keywords:

*CHANGE_OPTION
*CONTROL_DYNAMIC RELAXATION
*CONTROL_TERMINATION
*CONTROL_TIMESTEP

*DAMPING_GLOBAL
*DATABASE_OPTION
*DATABASE_BINARY_OPTION
*DELETE_OPTION
*INTERFACE_SPRINGBACK
*RIGID_DEFORMABLE_OPTION
*TERMINATION_OPTION
*TITLE
*KEYWORD (see INTRODUCTION, Execution Syntax)
*CONTROL_CPU
*DEFINE_OPTION
*SET_OPTION

i.e., the keyword *STRESS_INITIALIZATION may not be used in the small restart. The user has to take care that nonphysical modifications to the input deck are avoided; otherwise, complete nonsense may be the result.

c) If many modifications are desired a so called full restart may be the appropriate choice. Then the keyword *STRESS_INITIALIZATION has to be provided in the input. As also outlined in the INTRODUCTION, Restart Analysis, either all parts can be initialized with the restart data or some selection of parts can be made for the stress initialization. See *STRESS_INITIALIZATION.

***CHANGE_OPTION**

Available options are:

BOUNDARY_CONDITION

CONTACT_SMALL_PENETRATION

CURVE_DEFINITION

RIGID_BODY_CONSTRAINT

RIGID_BODY_STOPPER

STATUS_REPORT_FREQUENCY

THERMAL_PARAMETERS

VELOCITY

VELOCITY_NODE

VELOCITY_RIGID_BODY

VELOCITY_ZERO

Purpose: Change some solution options.

RESTART

For **BOUNDARY_CONDITION** option define an arbitrary number of cards giving the nodal ID and the additional translational displacement boundary condition code. Previous boundary condition codes will continue to be imposed, i.e., a fixed node cannot be freed with this option. This input terminates when the next "*" card is encountered.

Card Format

	1	2	3	4	5	6	7	8
Variable	NID	BCC						
Type	I	I						

VARIABLE	DESCRIPTION
NID	Nodal point ID, see also *NODE.
BCC	New translational boundary condition code: EQ.1: constrained x displacement, EQ.2: constrained y displacement, EQ.3: constrained z displacement, EQ.4: constrained x and y displacements, EQ.5: constrained y and z displacements, EQ.6: constrained z and x displacements, EQ.7: constrained x, y, and z displacements.

For **CONTACT_SMALL_PENETRATION** option define an arbitrary number of cards giving a list of contact surface ID numbers where the small penetration check is to be turned on. This input terminates when the next "*" card is encountered. See the PENCHK variable on the *CONTACT definition.

Card Format

	1	2	3	4	5	6	7	8
Variable	ID1	ID2	ID3	ID4	ID5	ID6	ID7	ID8
Type	I	I	I	I	I	I	I	I

VARIABLE	DESCRIPTION
ID*n*	Contact ID for surface number *n*.

The **CURVE_DEFINITION** option allows a load curve to be redefined. *The new load curve must contain the same number of points as the curve it replaces.* The curve should be defined in the DEFINE_CURVE section of this manual. This input terminates when the next "*" card is encountered.

Card Format

	1	2	3	4	5	6	7	8
Variable	LCID							
Type	I							

VARIABLE	DESCRIPTION
LCID	Load curve ID

RESTART

The **RIGID_BODY_CONSTRAINT** option allows translational and rotational boundary conditions on a rigid body to be changed. This input terminates when the next "*" card is encountered. Also, see *CONSTRAINED_RIGID_BODIES.

Card Format

	1	2	3	4	5	6	7	8
Variable	PID	TC	RT					
Type	I	I	I					

VARIABLE	DESCRIPTION
PID	Part ID, see *PART.
TC	Translational constraint: EQ.0: no constraints, EQ.1: constrained x displacement, EQ.2: constrained y displacement, EQ.3: constrained z displacement, EQ.4: constrained x and y displacements, EQ.5: constrained y and z displacements, EQ.6: constrained z and x displacements, EQ.7: constrained x, y, and z displacements.
RC	Rotational constraint: EQ.0: no constraints, EQ.1: constrained x rotation, EQ.2: constrained y rotation, EQ.3: constrained z rotation, EQ.4: constrained x and y rotations, EQ.5: constrained y and z rotations, EQ.6: constrained z and x rotations, EQ.7: constrained x, y, and z rotations.

The **RIGID_BODY_STOPPER** option allows existing stoppers to be redefined. This input terminates when the next "*" card is encountered. See *CONSTRAINED_RIGID_BODY_ STOPPERS.

New stopper definitions cannot be introduced in this section. Existing stoppers can be modified.

Card Formats

Card 1

Variable	PID	LCMAX	LCMIN	PSIDMX	PSIDMN	LCVMNX	DIR	VID
Type	I	I	I	I	I	I	I	I
Default	required	0	0	0	0	0	required	0

Card 2

Variable	BIRTH	DEATH						
Type	F	F						
Default	0	10^{28}						

VARIABLE	DESCRIPTION
PID	Part ID of master rigid body, see *PART.
LCMAX	Load curve ID defining the maximum coordinate as a function of time: EQ.0: no limitation of the maximum displacement. New curves can be defined by the *DEFINE_CURVE within the present restart deck.
LCMIN	Load curve ID defining the minimum coordinate as a function of time: EQ.0: no limitation of the minimum displacement. New curves can be defined by the *DEFINE_CURVE within the present restart deck.
PSIDMX	Optional part set ID of rigid bodies that are slaved in the maximum coordinate direction to the master rigid body. This option requires additional input by the *SET_PART definition.

VARIABLE	DESCRIPTION
PSIDMN	Optional part set ID of rigid bodies that are slaved in the minimum coordinate direction to the master rigid body. This option requires additional input by the *SET_PART definition.
LCVMNX	Load curve ID which defines the maximum absolute value of the velocity that is allowed within the stopper: EQ.0: no limitation of the minimum displacement.
DIR	Direction stopper acts in: EQ.1: x-translation, EQ.2: y-translation, EQ.3: z-translation, EQ.4: arbitrary, defined by vector VID, EQ.5: x-axis rotation, EQ.6: y-axis rotation, EQ.7: z-axis rotation, EQ.8: arbitrary, defined by vector VID.
VID	Vector for arbitrary orientation of stopper. The vector must be defined by a *DEFINE_VECTOR within the present restart deck.
BIRTH	Time at which stopper is activated.
DEATH	Time at which stopper is deactivated.

The optional definition of part sets in minimum or maximum coordinate directions allows the motion to be controlled in an arbitrary direction.

The **STATUS_REPORT_FREQUENCY** option allows the output status interval to be changed.

Card Format

	1	2	3	4	5	6	7	8
Variable	IKEDIT							
Type	I							

VARIABLE	DESCRIPTION
IKEDIT	Problem status report interval steps in the D3HSP output file: EQ.0: interval remains unchanged.

RESTART

The **VELOCITY** option allows a new velocity field to be imposed at restart. Termination of this input is when the next "*" card is read.

Card Format

Card 1	1	2	3	4	5	6	7	8
Variable	NSID							
Type	I							
Default	none							
Remark	1							

Card 2	1	2	3	4	5	6	7	8
Variable	VX	VY	VZ	VXR	VYR	VZR		
Type	F	F	F	F	F	F		
Default	0.	0.	0.	0.	0.	0.		

VARIABLE	DESCRIPTION
NSID	Nodal set ID containing nodes for initial velocity.
VX	Velocity in x-direction.
VY	Velocity in y-direction.
VZ	Velocity in z-direction.

VARIABLE	DESCRIPTION
VXR	Rotational velocity about the x-axis.
VYR	Rotational velocity about the y-axis.
VZR	Rotational velocity about the z-axis.

Remarks:

1. If a node is initialized on more than one input card set, then the last set input will determine its velocity, unless it is specified on a *CHANGE_VELOCITY_NODE card.
2. Undefined nodes will have their nodal velocities set to zero if a *CHANGE_VELOCITY definition is encountered in the restart deck.
3. If both *CHANGE_VELOCITY and *CHANGE_VELOCITY_ZERO cards are defined then all velocities will be reset to zero.

RESTART

The **THERMAL_PARAMETERS** option allows parameters used by a thermal or coupled structural/thermal analysis to be changed. These parameters were initially defined on the *CONTROL_THERMAL cards. Two cards are defined for this option.

Card Format (Card 1 of 2)

	1	2	3	4	5	6	7	8
Variable	TS	DT	TMIN	TMAX	DTEMP	TSCP		
Type	I	F	F	F	F	F		

Card Format (Card 2 of 2)

	1	2	3	4	5	6	7	8
Variable	REFMAX	TOL						
Type	I	F						

VARIABLE	DESCRIPTION
TS	Thermal time step code: EQ.0: No change, EQ.1: Fixed timestep, EQ.2: variable timestep.
DT	Thermal time step on restart: EQ.0: No change.
TMIN	Minimum thermal timestep: EQ.0: No change.
TMAX	Maximum thermal timestep: EQ.0: No change.
DTEMP	Maximum temperature change in a thermal timestep: EQ.0: No change.
TSCP	Time step control parameter (0.0 < TSCP < 1.0): EQ.0: No change.
REFMAX	Maximum number of reformations per thermal time step: EQ.0: No change.

VARIABLE	DESCRIPTION
TOL	Non-linear convergence tolerance: EQ.0: No change.

The **VELOCITY_NODE** option allows the velocity of nodal points to be changed at restart. Termination of this input is when the next "*" card is read.

Card Format

	1	2	3	4	5	6	7	8
Variable	NID	VX	VY	VZ	VXR	VYR	VZR	
Type	I	F	F	F	F	F	F	
Default	none	0.	0.	0.	0.	0.	0.	

VARIABLE	DESCRIPTION
NID	Node ID
VX	Translational velocity in x-direction.
VY	Translational velocity in y-direction.
VZ	Translational velocity in z-direction.
VXR	Rotational velocity about the x-axis.
VYR	Rotational velocity about the y-axis.
VZR	Rotational velocity about the z-axis.

Remarks:

1. If a node is initialized on more than one input card set, then the last set input will determine its velocity, unless it is specified on a *CHANGE_VELOCITY_NODE card.
2. Undefined nodes will have their nodal velocities set to zero if a *CHANGE_VELOCITY definition is encountered in the restart deck.
3. If both *CHANGE_VELOCITY and *CHANGE_VELOCITY_ZERO cards are defined then all velocities will be reset to zero.

The **VELOCITY_RIGID_BODY** option allows the velocity components of a rigid body to be changed at restart. Termination of this input is when the next "*" card is read.

Card Format

	1	2	3	4	5	6	7	8
Variable	PID	VX	VY	VZ	VXR	VYR	VZR	
Type	I	F	F	F	F	F	F	
Default	none	0.	0.	0.	0.	0.	0.	

VARIABLE	DESCRIPTION
PID	Part ID of rigid body.
VX	Translational velocity in x-direction.
VY	Translational velocity in y-direction.
VZ	Translational velocity in z-direction.
VXR	Rotational velocity about the x-axis.
VYR	Rotational velocity about the y-axis.
VZR	Rotational velocity about the z-axis.

Remarks:

1. Rotational velocities are defined about the center of mass of the rigid body.
2. Rigid bodies not defined in this section will not have their velocities modified.

The **VELOCITY_ZERO** option resets the velocities to zero at the start of the restart. Only the *CHANGE_VELOCITY_ZERO card is required for this option without any further input.

***CONTROL_DYNAMIC_RELAXATION**

Purpose: Define controls for dynamic relaxation.

Card Format

	1	2	3	4	5	6	7	8
Variable	NRCYCK	DRTOL	DRFCTR	DRTERM	TSSFDR	IRELAL	EDTTL	IDRFLG
Type	I	F	F	F	F	I	F	I
Default	250	0.001	0.995	infinity	TSSFAC	0	0.0	0
Remarks	1	1	1	1	1			1

VARIABLE	DESCRIPTION
NRCYCK	Number of iterations between convergence checks, for dynamic relaxation option (default = 250).
DRTOL	Convergence tolerance for dynamic relaxation option (default = 0.001).
DRFCTR	Dynamic relaxation factor (default = .995).
DRTERM	Optional termination time for dynamic relaxation. Termination occurs at this time or when convergence is attained (default = infinity).
TSSFDR	Scale factor for computed time step during dynamic relaxation. If zero, the value is set to TSSFAC defined on *CONTROL_TERMINATION. After converging, the scale factor is reset to TSSFAC.
IRELAL	Automatic control for dynamic relaxation option based on algorithm of Papadrakakis [Papadrakakis 1981].
EDTTL	Convergence tolerance on automatic control of dynamic relaxation.
IDRFLG	Dynamic relaxation flag for stress initialization: EQ.0: not active, EQ.1: dynamic relaxation is activated.

Remark:

1. If a dynamic relaxation relaxation analysis is being restarted at a point before convergence was obtained, then NRCYCK, DRTOL, DRFCTR, DRTERM and TSSFDR will default to their previous values, and IDRFLG will be set to 1.

2. If dynamic relaxation is activated after a restart from a normal transient analysis LS-DYNA3D continues the output of data as it would without the dynamic relaxation being active. This is unlike the dynamic relaxation phase at the beginning of the calculation when a separate database is not used. Only load curves that are flagged for dynamic relaxation are applied after restarting.

***CONTROL_TERMINATION**

Purpose: Stop the job.

Card Format

	1	2	3	4	5	6	7	8
Variable	ENDTIM	ENDCYC						
Type	F	I						

VARIABLE	DESCRIPTION
ENDTIM	Termination time: EQ:0.0 Termination time remains unchanged.
ENDCYC	Termination cycle. The termination cycle is optional and will be used if the specified cycle is reached before the termination time. EQ:0.0 Termination cycle remains unchanged.

This is a reduced version of the *CONTROL_TERMINATION card used in the initial input deck.

RESTART

*CONTROL_TIMESTEP

Purpose: Set time step size control using different options.

Card Format

	1	2	3	4	5	6	7	8
Variable	DUMMY	TSSFAC	ISDO	DUMMY	DT2MS	LCTM		
Type	F	F	I	F	F	I		

VARIABLE	DESCRIPTION
DUMMY	Dummy field, see remark 1 below.
TSSFAC	Scale factor for computed time step. EQ:0.0. TSSFAC remains unchanged.
ISDO	Basis of time size calculation for 4-node shell elements, ISDO 3-node shells use the shortest altitude for options 0,1 and the shortest side for option 2. This option has no relevance to solid elements, which use a length based on the element volume divided by the largest surface area: EQ.0: characteristic length=area/(longest side), EQ.1: characteristic length=area/(longest diagonal), EQ.2: based on bar wave speed and MAX **[**shortest side, area/longest side**]**. THIS LAST OPTION CAN GIVE A MUCH LARGER TIME STEP SIZE THAT CAN LEAD TO INSTABILITIES IN SOME APPLICATIONS, ESPECIALLY WHEN TRIANGULAR ELEMENTS ARE USED.
DUMMY	Dummy field, see remark 1 below.
DT2MS	New time step for mass scaled calculations. Mass scaling must be active in the time zero analysis. EQ:0.0. DT2MS remains unchanged.
LCTM	Load curve ID that limits maximum time step size: EQ:0. LCTM remains unchanged.

Remark:

1. This a reduced version of the *CONTROL_TIMESTEP used in the initial analysis. The dummy fields are included to maintain compatability. If using free format input then a 0.0 should be entered for the dummy values.

***DAMPING_GLOBAL**

Purpose: Define mass weigthed nodal damping that applies globally to the deformable nodes.

Card Format

	1	2	3	4	5	6	7	8
Variable	LCID	VALDMP						
Type	I	F						
Default	0	0.0						

VARIABLE	DESCRIPTION
LCID	Load curve ID which specifies node system damping: EQ.n: system damping is given by load curve n. The damping force applied to each node is f=-d(t) mv, where d(t) is defined by load curve n.
VALDMP	System damping constant, d (this option is bypassed if the load curve number defined above is nonzero).

RESTART

***DATABASE_OPTION**

Options for ASCII files include. If a file is not specified in the restart deck then the output interval for the file will remain unchanged.

SECFORC	Cross section forces.
RWFORC	Wall forces.
NODOUT	Nodal point data.
ELOUT	Element data.
GLSTAT	Global data.
DEFORC	Discrete elements.
MATSUM	Material energies.
NCFORC	Nodal interface forces.
RCFORC	Resultant interface forces.
DEFGEO	Deformed geometry file
SPCFORC	Set dt for spc reaction forces.
SWFORC	Nodal constraint reaction forces (spotwelds and rivets).
ABSTAT	Set dt for airbag statistics.
NODFOR	Set dt for nodal force groups.
BNDOUT	Boundary condition forces and energy
RBDOUT	Set dt for rigid body data.
GCEOUT	Set dt for geometric contact entities.
SLEOUT	Set dt for sliding interface energy.
JNTFORC	Set dt for joint force file.
SBTOUT	Set dt for seat belt output file.
AVSFLT	Set dt for AVS database.
MOVIE	Set dt for MOVIE.
MPGS	Set dt for MPGS.
TPRINT	Set dt for thermal file.

Card Format

	1	2	3	4	5	6	7	8
Variable	DT							
Type	F							

VARIABLE	DESCRIPTION
DT	Time interval between outputs: EQ:0.0 output interval is unchanged.

To terminate output to a particular file set DT to a high value.

RESTART

***DATABASE_BINARY_OPTION**

Options for binary output files with the default names given include:

D3PLOT	Dt for complete output states.
D3THDT	Dt for time history data for element subsets.
D3DUMP	Binary output restart files. Define output frequency in cycles
RUNRSF	Binary output restart file. Define output frequency in cycles.
INTFOR	Dt for contact surface Interface database.

Card Format

	1	2	3	4	5	6	7	8
Variable	DT/CYCL							
Type	F							

VARIABLE	DESCRIPTION
DT	Time interval between outputs. EQ:0.0. Time interval remains unchanged.
CYCL	Output interval in time steps. EQ:0.0. output interval remains unchanged.

***DELETE_OPTION**

Available options are:

CONTACT
ENTITY
PART
ELEMENT_BEAM
ELEMENT_SHELL
ELEMENT_SOLID
ELEMENT_TSHELL

Purpose: Delete contact surfaces, parts, or elements by a list of IDs.

For **CONTACT**, **ENTITY**, or **PART** option.

Card Format

	1	2	3	4	5	6	7	8
Variable	ID1	ID2	ID3	ID4	ID5	ID6	ID7	ID8
Type	I	I	I	I	I	I	I	I

VARIABLE	DESCRIPTION
IDI	Contact ID/Part ID

For *DELETE_CONTACT a negative ID implies that the absoulute value gives the contact surface which is to be activated

For the four **ELEMENT** options. Termination of input is when the next "*" card is read.

Card Format

	1	2	3	4	5	6	7	8
Variable	ESID							
Type	I							

VARIABLE	DESCRIPTION
ESID	Element set ID, see ***SET_SOLID, *SET_BEAM, *SET_SHELL, *SET_TSHELL**.

***INTERFACE_SPRINGBACK**

Purpose: Define a material subset for an implicit springback calculation in LS-NIKE3D and any nodal constraints to eliminate rigid body degrees-of-freedom. Generally, only the materials that make up the original blank are included in the springback calculation. After termination of the LS-DYNA3D computation, an input deck for LS-NIKE3D and a stress initialization file for LS-NIKE3D are written.

Card Format

	1	2	3	4	5	6	7	8
Variable	PSID							
Type	I							

VARIABLE	DESCRIPTION
PSID	Part set ID for springback, see *SET_PART.

Define a list of nodal points that are constrained for the springback. This section is terminated by an "*" indicating the next input section.

Card Format

	1	2	3	4	5	6	7	8
Variable	NID	TC	RC					
Type	I	F	F					
Default	none	0.	0.					

VARIABLE	DESCRIPTION
NID	Node ID

VARIABLE	DESCRIPTION
TC	Tranlational constraint: EQ.0: no constraints, EQ.1: constrained x displacement, EQ.2: constrained y displacement, EQ.3: constrained z displacement, EQ.4: constrained x and y displacements, EQ.5: constrained y and z displacements, EQ.6: constrained z and x displacements, EQ.7: constrained x, y, and z displacements.
RC	Rotational constraint: EQ.0: no constraints, EQ.1: constrained x rotation, EQ.2: constrained y rotation, EQ.3: constrained z rotation, EQ.4: constrained x and y rotations, EQ.5: constrained y and z rotations, EQ.6: constrained z and x rotations, EQ.7: constrained x, y, and z rotations.

RIGID_DEFORMABLE_*OPTION

The OPTIONS available are:

CONTROL	
D2D	(Deformable to rigid part switch)
R2D	(Rigid to deformable part switch)

Purpose: Define parts to be switched from rigid to deformable and deformable to rigid in a restart. It is only possible to switch parts on a restart if part switching was activated in the time zero analysis. See *CONTROL_DEFORMABLE_RIGID for details of part switching.

RESTART

For the **CONTROL** option define the following card:

Card Format

	1	2	3	4	5	6	7	8
Variable	NRBF	NCSF	RWF	DTMAX				
Type	I	I	I	F				
Default	0	0	0	none				

VARIABLE	DESCRIPTION
NRBF	Flag to delete or activate nodal rigid bodies. If nodal rigid bodies or generalized, weld definitions are active in the deformable bodies that are switched to rigid, then the definitions should be deleted to avoid instabilities: EQ.0: no change, EQ.1: delete, EQ.2: activate.
NCSF	Flag to delete or activate nodal constraint set. If nodal constraint/spotweld definitions are active in the deformable bodies that are switched to rigid, then the definitions should be deleted to avoid instabilities: EQ.0: no change, EQ.1: delete, EQ.2: activate.
RWF	Flag to delete or activate rigid walls: EQ.0: no change, EQ.1: delete, EQ.2: activate.
DTMAX	Maximum permitted time step size after restart.

For the **D2R** option define the following card. Termination of this input is when the next "*" card is read.

Card Format

	1	2	3	4	5	6	7	8
Variable	PID	MRB						
Type	I	I						
Default	none	0						

VARIABLE	DESCRIPTION
PID	Part ID of the part which is switched to a rigid material.
MRB	Part ID of the master rigid body to which the part is merged. If zero, the part becomes either an independent or master rigid body.

For the **R2D** option define the following card. Termination of this input is when the next "*" card is read.

Card Format

	1	2	3	4	5	6	7	8
Variable	PID							
Type	I							
Default	none							

VARIABLE	DESCRIPTION
PID	Part ID of the part which is switched to a deformable material.

RESTART

***STRESS_INITIALIZATION_OPTION**

This keyword allows a full deck restart to be performed in LS-DYNA3D. For a full deck restart a complete input deck has to be included in the restart deck. The stress initialization feature allows all or a number of parts to be initialized on restart.

The options that are available with this keyword are:

<BLANK>
DISCRETE
SEATBELTS

***STRESS_INITIALIZATION**

If this card is specified without further input then all parts in the new analysis are initialized from the corresponding part of the old analysis. Further all seatbelt and discrete parts are initialized.

If only a subset of parts are to be initialized in the new analysis then define as many of the following cards as necessary. Termination of this input is when the next "*" card is read.

Card Format

Card 1	1	2	3	4	5	6	7	8
Variable	PIDO	PIDN						
Type	I	I						
Default	none	PIDO						

VARIABLE	DESCRIPTION
PIDO	Old part ID, see *PART.
PIDN	New part ID, see *PART: EQ:0. New part ID is the same as the old part ID.

If one or more of the above cards are defined then discrete and and seatbelt elements will not be initialized unless the additional option cards *STRESS_INITIALIZATION_DISCRETE and *STRESS_INITIALIZATION_SEATBELT are defined.

***STRESS_INITIALIZATION_DISCRETE**

Initialize all discrete parts from the old parts. No further input is required with this card. This card is not required if *STRESS_INITIALIZATION is specified without further input.

***STRESS_INITIALIZATION_SEATBELT**

Initialize all seatbelt parts from the old parts. No further input is required with this card. This card is not required if *STRESS_INITIALIZATION is specified without further input.

TERMINATION_*OPTION

Purpose: Stop the job depending on some displacement conditions.

Available options include:

NODE
BODY

Caution: The inputs are different for the nodal and rigid body stop conditions. The nodal stop condition works on the global coordinate position, while the body stop condition works on the relative global translation. The number of termination conditions cannot exceed the maximum of 10 or the number specified in the original analysis.

The analysis terminates for *TERMINATION_NODE when the current position of the node specified reaches either the maximum or minimum value (stops 1, 2 or 3), or picks up force from any contact surface (stop 4). For *TERMINATION_BODY the analysis terminates when the center of mass displacement of the rigid body specified reaches either the maximum or minimum value (stops 1, 2 or 3) or the displacement magnitude of the center of mass is exceeded (stop 4). If more than one condition is input, the analysis stops when any of the conditions is satisfied. ***This input completely overides the existing termination conditions defined in the time zero run.***

Termination by other means is controlled by the *CONTROL_TERMINATION control card.

For both options, the input is identical:
Card Format

	1	2	3	4	5	6	7	8
Variable	NID/PID	STOP	MAXC	MINC				
Type	I	I	F	F				
Default	none	none	-	-				

For the **NODE** option:

VARIABLE	DESCRIPTION
NID	Node ID
STOP	Stop criterion: EQ.1: global x direction, EQ.2: global y direction, EQ.3: global z direction, EQ.4: stop if node touches contact surface.
MAXC	Maximum (most positive) coordinate, options 1, 2 and 3 above only.
MINC	Minimum (most negative) coordinate, options 1, 2 and 3 above only.

For the **BODY** option:

VARIABLE	DESCRIPTION
PID	Part ID of rigid body
STOP	Stop criterion: EQ.1: global x direction, EQ.2: global y direction, EQ.3: global z direction, EQ.4: stop if displacement magnitude is exceeded.
MAXC	Maximum (most positive) displacement, options 1, 2, 3 and 4: EQ:0.0. MAXC set to 1.0e21
MINC	Minimum (most negative) displacement, options 1, 2 and 3 above only: EQ:0.0. MINC set to -1.0e21

***TITLE**

Purpose: Define job title.

Card Format

	1	2	3	4	5	6	7	8
Variable	TITLE							
Type	C							
Default	LS-DYNA3D USER INPUT							

VARIABLE	DESCRIPTION
TITLE	Heading to appear on output.

REFERENCES

Allman, D.J., "A Compatible Triangular Element Including Vertex Rotations for Plane Elasticity Analysis," Comp. Struct., 19,1-8, (1984).

Bammann, D.J. and E.C. Aifantis, "A Model for Finite-Deformation Plasticity," Acta Mechanica, 70, 1-13 (1987).

Bammann, D.J. and G. Johnson, "On the Kinematics of Finite-Deformation Plasticity," Acta Mechanica, *69*, 97-117 (1987).

Bammann, D.J., "Modeling the Temperature and Strain Rate Dependent Large Deformation of Metals," Proceedings of the 11th US National Congress of Applied Mechanics, Tuscon, AZ, (1989).

Bammann, D.J., M.L. Chiesa, A. McDonald, W.A. Kawahara, J.J. Dike, and V.D. Revelli, "Predictions of Ductile Failure in Metal Structures," in AMD-Vol. 107, Failure Criteria and Analysis in Dynamic Response, Edited by. H.E. Lindberg, 7-12, (1990).

Bandak, F.A., private communications, U.S. Dept. of Trans., Division of Biomechanics Research, 400 7th St., S.W. Washington, D.C. 20590 (1991).

Barlat, F. and J. Lian, "Plastic Behavior and Stretchability of Sheet Metals. Part I: A Yield Function for Orthotropic Sheets Under Plane Stress Conditions," Int. J. of Plasticity, Vol. 5, pp. 51-66 (1989).

Barlat, F., D.J. Lege, and J.C. Brem, "A Six-Component Yield Function for Anisotropic Materials," Int. J. of Plasticity, 7, 693-712, (1991).

Bazeley, G.P., W.K. Cheung, R.M. Irons, and O.C. Zienkiewicz, "Triangular Elements in Plate Bending-Confirming and Nonconforming Solutions in Matrix Methods and Structural Mechanics," Proc. Conf. on Matrix Methods in Structural Analysis, Rept. AFFDL-R-66-80, Wright Patterson AFB, 547-576 (1965).

Belytschko, T.B. and A.H. Marchertas, "Nonlinear Finite Element Method for Plates and its Application to the Dynamic Response of Reactor Fuel Subassemblies," Trans, ASME J. Pressure Vessel Tech., 251-257 (1974).

Belytschko, T.B. and C.S. Tsay, "Explicit Algorithms for Nonlinear Dynamics of Shells," AMD-Vol.48, ASME, 209-231 (1981).

Belytschko, T.B. and C.S. Tsay, "Explicit Algorithms for Nonlinear Dynamics of Shells," Comp. Meth. Appl. Mech. Eng., 43, 251-276, (1984).

Belytschko, T.B. and C.S. Tsay, "A Stabilization Procedure for the Quadrilateral Plate Element with One-Point Quadrature," Int. J. Num. Method. Eng., 19, 405-419 (1983).

Belytschko, T.B., H. Stolarski, and N. Carpenter, "A C^o Triangular Plate Element with One-Point Quadrature," Int. J. Num. Meth. Eng., 20, 787-802 (1984).

REFERENCES

Belytschko, T.B., L. Schwer, and M.J. Klein, "Large Displacement Transient Analysis of Space Frames," Int. J. Num. Eng., 11, 65-84 (1977).

Benson, D.J. and J.O. Hallquist, "A Simple Rigid Body Algorithm for Structural Dynamics Programs," Int. J. Numer. Meth. Eng., 22, (1986).

Benson, D.J. and J.O. Hallquist, "A Single Surface Contact Algorithm for the Postbuckling Analysis of Shell Structures," Comp. Meths. Appl. Mech. Eng., 78, 141-163 (1990).

Bilkhu, S.S., M. Founas, and G.S. Nasholtz, "Material Modeling of Structural Foams in Finite Element Analysis Using Compressive Uniaxial and Triaxial Data," SAE (Nat. Conf.) Detroit 1993, pp. 4-34.

Brode, H.L., "Height of Burst Effects at High Overpressure," RAND, RM-6301-DASA, DASA 2506, (1970).

Brown, B.E. and J.O. Hallquist, "TAURUS: An Interactive Post-Processor for the Analysis Codes NIKE3D, DYNA3D, TACO3D, and GEMINI," University of California, Lawrence Livermore National Laboratory, Rept. UCID-19392 (1982) Rev. 1 (1984).

Burton, D.E. et al. "Physics and Numerics of the TENSOR Code," Lawrence Livermore National Laboratory, Internal Document UCID-19428, (July 1982).

Chang, F.K. and K.Y. Chang, "A Progressive Damage Model for Laminated Composites Containing Stress Concentration," J. of Composite Materials, 21, 834-855 (1987a).

Chang, F.K. and K.Y. Chang, "Post-Failure Analysis of Bolted Composite Joints in Tension or Shear-Out Mode Failure," J. of Composite Materials, 21 809-833 (1987b).

Chung, K. and K. Shah, "Finite Element Simulation of Sheet Metal Forming for Planar Anisotropic Metals," Int. J. of Plasticity, 8, 453-476, (1992).

Cochran, S.G. and J. Chan, "Shock Initiation and Detonation Models in One and Two Dimensions," University of California, Lawrence Livermore National Laboratory, Rept. UCID-18024 (1979).

Couch, R., E. Albright, and N. Alexander, "The Joy Computer Code," Lawrence Livermore National Laboratory, Internal Document Rept. UCID-19688, (January, 1983).

CRAY-1 Computer System CFT Reference Manual, Cray Research Incorporated, Bloomington, NM., Publication No. 2240009 (1978).

DeRuntz, J.A. Jr., "Reference Material for USA, The Underwater Shock Analysis Code, USA-STAGS, and USA-STAGS-CFA," Report LMSC-P032568, Computational Mechanics Laboratory, Lockheed Palo Alto Research Laboratory, Palo Alto, CA. (1993).

Dobratz, B.M., "LLNL Explosives Handbook, Properties of Chemical Explosives and Explosive Simulants," University of California, Lawrence Livermore National Laboratory, Rept. UCRL-52997 (1981).

REFERENCES

Englemann, B. E., R.G. Whirley, and G.L. Goudreau, "A Simple Shell Element Formulation for Large-Scale Elastoplastic Analysis," CED-Vol. 3. Analytical and Computational Models of Shells, A.K. Noor, T. Belytschko, and J.C. Simo, Editors, 1989, pp. 399-416.

Flanagan, D.P. and T. Belytschko, "A Uniform Strain Hexahedron and Quadrilateral and Orthogonal Hourglass Control," Int. J. Numer. Meths. Eng., 17, 679-706 (1981)

Ginsberg, M. and J. Johnson, "Benchmarking the Performance of Physical Impact Simulation Software on Vector and Parallel Computers," Applications Track of Supercomputing, IEEE monograph, Computer Society Press, March, 1989.

Giroux, E.D. "HEMP User's Manual," University of California, Lawrence Livermore National Laboratory, Rept. UCRL-51079 (1973).

Goudreau, G.L. and J.O. Hallquist, "Recent Developments in Large Scale Finite Element Lagrangian Hydrocode Technology," J. Comp. Meths. Appl. Mechs. Eng., 30 (1982).

Hallquist, J.O., "Preliminary User's Manuals for DYNA3D and DYNAP (Nonlinear Dynamic Analysis of Solids in Three Dimension)," University of California, Lawrence Livermore National Laboratory, Rept. UCID-17268 (1976) and Rev. 1 (1979).[**a**]

Hallquist, J.O., "A Procedure for the Solution of Finite Deformation Contact-Impact Problems by the Finite Element Method," University of California, Lawrence Livermore National Laboratory, Rept. UCRL-52066 (1976).

Hallquist, J.O., "A Numerical Procedure for Three-Dimensional Impact Problems," American Society of Civil Engineering, Preprint 2956 (1977).

Hallquist, J.O., "A Numerical Treatment of Sliding Interfaces and Impact," in: K.C. Park and D.K. Gartling (eds.) Computational Techniques for Interface Problems, AMD Vol. 30, ASME, New York (1978).

Hallquist, J.O., "NIKE2D: An Implicit, Finite-Element Code for Analyzing the Static and Dynamic Response of Two-Dimensional Solids," University of California, Lawrence Livermore National Laboratory, Rept. UCRL-52678 (1979).[**b**]

Hallquist, J.O., "User's Manual for DYNA2D – An Explicit Two-Dimensional Hydrodynamic Finite Element Code with Interactive Rezoning," University of California, Lawrence Livermore National Laboratory, Rept. UCID-18756 (1980).

Hallquist, J.O., "User's Manual for DYNA3D and DYNAP (Nonlinear Dynamic Analysis of Solids in Three Dimensions)," University of California, Lawrence Livermore National Laboratory, Rept. UCID-19156 (1981).[**a**]

Hallquist, J. O., "NIKE3D: An Implicit, Finite-Deformation, Finite-Element Code for Analyzing the Static and Dynamic Response of Three-Dimensional Solids," University of California, Lawrence Livermore National Laboratory, Rept. UCID-18822 (1981).[**b**]

REFERENCES

Hallquist, J.O., "DYNA3D User's Manual (Nonlinear Dynamic Analysis of Solids in Three Dimensions)," University of California, Lawrence Livermore National Laboratory, Rept. UCID-19156 (1982; Rev. 1: 1984; Rev. 2: 1986).

Hallquist, J.O., "Theoretical Manual for DYNA3D," University of California, Lawrence Livermore National Laboratory, Rept. UCID-19401 (March, 1983).

Hallquist, J.O., "DYNA3D User's Manual (Nonlinear Dynamic Analysis of Solids in Three Dimensions)," University of California, Lawrence Livermore National Laboratory, Rept. UCID-19156 (1988, Rev. 4).

Hallquist, J.O., "LS-DYNA3D User's Manual (Nonlinear Dynamic Analysis of Solids in Three Dimensions)," Livermore Software Technology Corporation, Rept. 1007 (1990).

Hallquist, J.O., D.J. Benson, and G.L. Goudreau, "Implementation of a Modified Hughes-Liu Shell into a Fully Vectorized Explicit Finite Element Code," Proceedings of the International Symposium on Finite Element Methods for Nonlinear Problems, University of Trondheim, Trondheim, Norway (1985).

Hallquist, J.O. and D.J. Benson, "A Comparison of an Implicit and Explicit Implementation of the Hughes-Liu Shell," Finite Element Methods for Plate and Shell Structures, T.J.R. Hughes and E. Hinton, Editors, 394-431, Pineridge Press Int., Swanea, U.K. (1986).

Hallquist, J.O. and D.J. Benson, "DYNA3D User's Manual (Nonlinear Dynamic Analysis of Solids in Three Dimensions)," University of California, Lawrence Livermore National Laboratory, Rept. UCID-19156 (Rev. 2: 1986; Rev. 3: 1987).

Hallquist, J.O., D.W. Stillman, T.J.R. Hughes, C. and Tarver,"Modeling of Airbags Using MVMA/DYNA3D," LSTC Report (1990).

Herrmann, L.R. and F.E. Peterson, "A Numerical Procedure for Viscoelastic Stress Analysis," Seventh Meeting of ICRPG Mechanical Behavior Working Group, Orlando, FL, CPIA Publication No. 177, 1968.

Hill, R., "A Theory of the Yielding and Plastic Flow of Anisotropic Metals," Proceedings of the Royal Society of London, Series A., Vol. 193, 1948, pp. 281-197.

Hill, R., "Constitutive Modelling of Orthotropic Plasticity in Sheet Metals," J. Mech. Phys. Solids, Vol. 38, No. 3, 1989, pp. 405-417.

Hughes, T.J.R. and W.K. Liu, "Nonlinear Finite Element Analysis of Shells: Part I. Three-Dimensional Shells." Comp. Meths. Appl. Mechs., 27, 331-362 (1981a).

Hughes, T.J.R. and W.K. Liu, "Nonlinear Finite Element Analysis of Shells: Part II. Two-Dimensional Shells." Comp. Meths. Appl. Mechs., 27, 167-181 (1981b).

Hughes, T.J.R., W.K. Liu, and I. Levit, "Nonlinear Dynamics Finite Element Analysis of Shells." Nonlinear Finite Element Analysis in Struct. Mech., Eds. W. Wunderlich, E. Stein, and K.J. Bathe, Springer-Verlag, Berlin, 151- 168 (1981c).

Johnson, G.C. and D.J. Bammann, "A discussion of stress rates in finite deformation problems," Int. J. Solids Struct, 20, 725-737 (1984).

Johnson, G.R. and W.H. Cook, "A Constitutive Model and Data for Metals Subjected to Large Strains, High Strain Rates and High Temperatures." Presented at the Seventh International Symposium on Ballistics, The Hague, The Netherlands, April 1983.

Kenchington, G.J., "A Non-Linear Elastic Material Model for DYNA3D," Proceedings of the DYNA3D Users Group Conference, September 1988, published by Boeing Computer Services (Europe) Limited.

Key, S.W. "HONDO – A Finite Element Computer Program for the Large Deformation Dynamic Response of Axisymmetric Solids," Sandia National Laboratories, Albuquerque, N.M., Rept. 74-0039 (1974).

Krieg, R.D. and S.W. Key, "Implementation of a Time Dependent Plasticity Theory into Structural Computer Programs," Vol. 20 of Constitutive Equations in Viscoplasticity: Computational and Engineering Aspects (American Society of Mechanical Engineers, New York, N.Y., 1976), pp. 125-137.

Lee, E.L. and C.M. Tarver, "Phenomenological Model of Shock Initiation in Heterogenous Explosives," PHYS. Fluids, Vol. 23, p. 2362 (1980).

MADYMO3D USER'S MANUAL, Version 4.3, TNO Road-Vehicles Research Institute, Department of Injury Prevention, The Hague, The Netherlands, (1990).

Maker, B.N., Private communication Lawrence Livermore National Laboratory, Dr. Maker programmed and implemented the compressible Mooney Rivlin rubber model (1987).

Matzenmiller, A. and J. K.Schm , "Crashworthiness Considerations of Composite Structures–A First Step with Explicit Time Integration in Nonlinear Computational Mechanics–State-of-the-Art," Ed. P. Wriggers, W. Wagner, Springer Verlay, , 1991.

Neilsen, M.K., H.S. Morgan, and R.D. Krieg, "A Phenomenological Constitutive Model for Low Density Polyurethane Foams," Rept. SAND86-2927, Sandia National Laboratories, Albuquerque, N.M., (1987)

Papadrakakis, M., "A Method for the Automatic Evaluation of the Dynamic Relaxation Parameters," Comp. Meth. Appl. Mech. Eng., Vol. 25, 1981, pp. 35-48.

Pelessone, D., Private communication, GA Technologies, P.O. Box 85608, San Diego, CA., Telephone No. 619-455-2501 (1986).

Sackett, S.J., "Geological/Concrete Model Development," Private Communication (1987).

Sandler, I.S. and D. Rubin, "An Algorithm and a Modular Subroutine for the Cap Model," Int. J. Numer. Analy. Meth. Geomech., 3, pp. 173-186 (1979).

Schwer, L.E., W. Cheva, and J.O. Hallquist, "A Simple Viscoelastic Model for Energy Absorbers Used in Vehicle-Barrier Impacts," in preparation.

REFERENCES

Simo, J.C., J.W. Ju, K.S. Pister, and R.L. Taylor, "An Assessment of the Cap Model: Consistent Return Algorithms and Rate-Dependent Extension," J. Eng. Mech., Vol. 114, No. 2, 191-218 (1988a).

Simo, J.C., J.W. Ju, K.S. Pister, and R.L. Taylor, "Softening Response, Completeness Condition, and Numerical Algorithms for the Cap Model," Int. J. Numer. Analy. Meth. Eng., (in press) (1988b).

Steinberg, D.J. and M.W. Guinan, "A High-Strain-Rate Constitutive Model for Metals," University of California, Lawrence Livermore National Laboratory, Rept. UCRL-80465 (1978).

Stillman, D.W. and J.O. Hallquist, "INGRID: A Three-Dimensional Mesh Generator for Modeling Nonlinear Systems," University of California, Lawrence Livermore National Laboratory, Rept. UCID-20506. (1985).

Storakers, B., "On Material Representation and Constitutive Branching in Finite Compressible Elasticity", Royal Institute of Technology, Stockholm, Sweden, (1985).

Stout, M.G., D.E. Helling, T.L. Martin, and G.R. Canova, Int. J. Plasticity, Vol. 1, pp. 163-174, 1985.

Taylor, L.M. and D.P. Flanagan, "PRONTO3D A Three-Dimensional Transient Solid Dynamics Program," Sandia Report: SAND87-1912, UC-32, (1989).

Tsai, S.W. and E.M. Wu, "A General Theory of Strength for Anisotropic Materials," J. Composite Materials, 5, 1971, pp. 73-96.

VDA Richtlinier (Surface Interfaces), Version 20, Verband der Automobilindustrie e.v., Frankfurt, Main, Germany, (1987).

Wang, J.T. and O.J. Nefske, "A New CAL3D Airbag Inflation Model," SAE paper 880654, 1988.

Whirley, R.G., J.O. Hallquist, and G.L. Goudreau, "An Assessment of Numerical Algorithms for Plane Stress and Shell Elastoplasticity on Supercomputers," Engineering Computations, Vol. 6, pp. 116-126, (1989).

Wilkins, M.L., R.E. Blum, E. Cronshagen, and P. Grantham, "A Method for Computer Simulation of Problems in Solid Mechanics and Gas Dynamics in Three Dimensions and Time," University of California, Lawrence Livermore National Laboratory, Rept. UCRL-51574 (1974).

Woodruff, J.P., "KOVEC User's Manual," University of California, Lawrence Livermore National Laboratory, Rept. UCRL-51079 (1973).

APPENDIX A: User Defined Materials

The addition of user material subroutine into LS-DYNA3D is relatively simple. A keyword *MAT_USER_DEFINED_MATERIAL_MODELS is required on which each user subroutine referenced. The number of history variables is arbitrary and can be any number greater than or equal to 0. The coordinate system definition is optional but is probably necessary if the model involves material that have directional properties such as composites and anisotropic plasticity models. When the coordinate system option is used, then all data passed to the constitutive model is in the local system. A bulk modulus and shear modulus are required for transmitting boundaries, contact interfaces, rigid body constraints, and time step size calculations. The number of constants read in columns 6-10 include the eight values for the coordinate system option if it is nonzero and two values for the bulk and shear modulus. Up to ten user subroutines can currently be implemented simultaneously to update the stresses in solids, shells, thick shells, and beam elements. A sample subroutine is given in this Appendix for treating an elastic material.

When implementing plane stress constitutive models for shells and beams, the strain increments in the directions of the zero normal stress must be determined. In shell elements this is the strain increment EPS(3) which is normal to the midsurface, and in beam elements this includes the strain increments EPS(2) and EPS(3) which are normal to the axis. These strain increments are used to account for thickness changes.

A sample subroutine is provided below for treating an elastic material.

```
      SUBROUTINE UMAT41 (CM,EPS,SIG,HISV,DT1,CAPA,ETYPE,TIME)
C************************************************************************
C |   LIVERMORE SOFTWARE TECHNOLOGY CORPORATION (LSTC)                    |
C |   ---------------------------------------------------------------    |
C |   COPYRIGHT 1987-1994, LSTC                                           |
C |   ALL RIGHTS RESERVED                                                 |
C************************************************************************
C
C     ISOTROPIC ELASTIC MATERIAL (SAMPLE USER SUBROUTINE)
C
C     VARIABLES
C
C     CM(1)=YOUNG'S MODULUS
C     CM(2)=POISSON'S RATIO
C
C     EPS(1)=LOCAL X STRAIN
C     EPS(2)=LOCAL Y STRAIN
C     EPS(3)=LOCAL Z STRAIN
C     EPS(4)=LOCAL XY STRAIN
C     EPS(5)=LOCAL YZ STRAIN
C     EPS(6)=LOCAL ZX STRAIN
C     EPS(1)=LOCAL X STRAIN
C
C     SIG(1)=LOCAL X STRESS
C     SIG(2)=LOCAL Y STRESS
C     SIG(3)=LOCAL Z STRESS
```

```
C     SIG(4)=LOCAL XY STRESS
C     SIG(5)=LOCAL YZ STRESS
C     SIG(6)=LOCAL ZX STRESS
C
C     HISV(1)=1ST HISTORY VARIABLE
C     HISV(2)=2ND HISTORY VARIABLE
C        .
C        .
C        .
C        .
C     HISV(N)=NTH HISTORY VARIABLE–SHALL NOT EXCEED VALUE GIVEN IN
C     *MAT_USER_DEFINED_MATERIAL_MODELS
C
C     DT1=CURRENT TIME STEP SIZE
C     CAPA=REDUCTION FACTOR FOR TRANSVERSE SHEAR
C     ETYPE:
C        EQ.“BRICK” FOR SOLID ELEMENTS
C        EQ.“SHELL” FOR ALL SHELL ELEMENTS
C        EQ.“BEAM”  FOR ALL BEAM ELEMENTS
C
C     TIME=CURRENT PROBLEM TIME.
C
C
C     ALL TRANSFORMATIONS INTO THE ELEMENT LOCAL SYSTEM ARE PERFORMED
C     PRIOR TO ENTERING THIS SUBROUTINE.  TRANSFORMATIONS BACK TO
C     THE GLOBAL SYSTEM ARE PERFORMED AFTER EXITING THIS SUBROUTINE.
C
C     ALL HISTORY VARIABLES ARE INITIALIZED TO ZERO IN THE INPUT PHASE.
C     INITIALIZATION OF HISTORY VARIABLES TO NONZERO VALUES MAY BE DONE
C     DURING THE FIRST CALL TO THIS SUBROUTINE FOR EACH ELEMENT.
C
C     ENERGY CALCULATIONS FOR THE DYNA3D ENERGY BALANCE ARE DONE
C     OUTSIDE THIS SUBROUTINE.
C
      CHARACTER*(*) ETYPE
      DIMENSION CM(*),EPS(*),SIG(*),HISV(*)
C
C     COMPUTE SHEAR MODULUS, G
C
      G2=CM(1)/(1.+CM(2))
      G =.5*G
C
      IF (ETYPE.EQ.‘BRICK’) THEN
      DAVG=(-EPS(1)-EPS(2)-EPS(3))/3.
      P=-DAVG*CM(1)/((1.-2.*CM(2)))
      SIG(1)=SIG(1)+P+G2*(EPS(1)+DAVG)
      SIG(2)=SIG(2)+P+G2*(EPS(2)+DAVG)
      SIG(3)=SIG(3)+P+G2*(EPS(3)+DAVG)
      SIG(4)=SIG(4)+P+EPS(4)
      SIG(5)=SIG(5)+P+EPS(5)
      SIG(6)=SIG(6)+P+EPS(6)
C
      ELSEIF (ETYPE.EQ. ‘SHELL’) THEN
C
      GC =CAPA*G
      Q1    =CM(1)*CM(2)/((1.0+CM(2))*(1.0-2.0*CM(2)))
      Q3    =1./(Q1+G2)
      EPS(3)=-Q1*(EPS(1)+EPS(2))*Q3
      DAVG  =(-EPS(1)-EPS(2)-EPS(3))/3.
      P     =-DAVG*CM(1)/((1.-2.*CM(2)))
      SIG(1)=SIG(1)+P+G2*(EPS(1)+DAVG)
```

```
      SIG(2)=SIG(2)+P+G2*(EPS(2)+DAVG)
      SIG(3)=0.0
      SIG(4)=SIG(4)+G *EPS(4)
      SIG(5)=SIG(5)+GC*EPS(5)
      SIG(6)=SIG(6)+GC*EPS(6)
C
      ELSEIF (ETYPE.EQ. 'BEAM') THEN
      Q1    =CM(1)*CM(2)/((1.0+CM(2))*(1.0-2.0*CM(2)))
      Q3    =Q1+2.0*G
      GC    =CAPA*G
      DETI  =1./(Q3*Q3-Q1*Q1)
      C22I  = Q3*DETI
      C23I  =-Q1*DETI
      FAC   =(C22I+C23I)*Q1
      EPS(2)=-EPS(1)*FAC-SIG(2)*C22I-SIG(3)*C23I
      EPS(3)=-EPS(1)*FAC-SIG(2)*C23I-SIG(3)*C22I
      DAVG  =(-EPS(1)-EPS(2)-EPS(3))/3.
      P     =-DAVG*CM(1)/(1.-2.*CM(2))
      SIG(1)=SIG(1)+P+G2*(EPS(1)+DAVG)
      SIG(2)=0.0
      SIG(3)=0.0
      SIG(4)=SIG(4)+GC*EPS(4)
      SIG(5)=0.0
      SIG(6)=SIG(6)+GC*EPS(6)
      ENDIF
C
      RETURN
      END
```

APPENDIX B: User Defined Airbag Sensor

The addition of a user sensor subroutine into LS-DYNA3D is relatively simple. The sensor is mounted on a rigid body which is attached to the structure. The motion of the sensor is provided in the local coordinate system defined for the rigid body in the definition of material model 20–the rigid material. When the user defined criterion is met for the deployment of the airbag, a flag is set and the deployment begins. All load curves relating to the mass flow rate versus time are then shifted by the initiation time. The user subroutine is given below with all the necessary information contained in the comment cards.

```
      SUBROUTINE AIRUSR (RBU,RBV,RBA,TIME,DT1,DT2,PARAM,HIST,ITRNON,
     . RBUG,RBVG,RBAG)
C*************************************************************
C |   LIVERMORE SOFTWARE TECHNOLOGY CORPORATION (LSTC)          |
C |   ------------------------------------------------------   |
C |   COPYRIGHT 1987, 1988, 1989 JOHN O. HALLQUIST, LSTC       |
C |   ALL RIGHTS RESERVED                                      |
C*************************************************************
C
C     USER SUBROUTINE TO INITIATE THE INFLATION OF THE AIRBAG
C
C     VARIABLES
C
C     DISPLACEMENTS ARE DEFINED AT TIME N+1 IN LOCAL SYSTEM
C     VELOCITIES ARE DEFINED AT TIME N+1/2 IN LOCAL SYSTEM
C     ACCELERATIONS ARE DEFINED AT TIME N IN LOCAL SYSTEM
C
C           RBU(1-3) TOTAL DISPLACEMENTS IN THE LOCAL XYZ DIRECTIONS
C           RBU(3-6) TOTAL ROTATIONS ABOUT THE LOCAL XYZ AXES
C           RBV(1-3) VELOCITIES IN THE LOCAL XYZ DIRECTIONS
C           RBV(3-6) ROTATIONAL VELOCITIES ABOUT THE LOCAL XYZ AXES
C           RBA(1-3) ACCELERATIONS IN THE LOCAL XYZ DIRECTIONS
C           RBA(3-6) ROTATIONAL ACCELERATIONS ABOUT THE LOCAL XYZ AXES
C           TIME IS THE CURRENT TIME
C           DT1 IS TIME STEP SIZE AT N-1/2
C           DT2 IS TIME STEP SIZE AT N+1/2
C           PARAM IS USER DEFINED INPUT PARAMETERS (MAX 25)
C           HIST IS USER DEFINED HISTORY VARIABLES (MAX 25)
C           ITRNON IS FLAG TO TURN ON THE AIRBAG INFLATION
C           RBUG,RBVG,RBAG, ARE SIMILAR TO RBU,RBV,RBA BUT ARE DEFINED
C           GLOBALLY.
C
C     THE USER SUBROUTINE SETS THE VARIABLE ITRNON TO:
C
C              ITRNON=0 BAG IS NOT INFLATED
C              ITRNON=1 BAG INFLATION BEGINS AND THIS SUBROUTINE IN NOT
C                       CALLED AGAIN
C
      DIMENSION RBU(6),RBV(6),PARAM(25),HIST(25),
     . RBUG(6),RBVG(6),RBAG(6)
      RETURN
      END
```

APPENDIX C: User Defined Solution Control

This subroutine may be provided by the user to control the I/O, monitor the energies and other solution norms of interest, and to shut down the problem whenever he pleases. The arguments are defined in the listing provided below. This subroutine is call each time step and does not need any control card to operate.

```
      SUBROUTINE UCTRL1 (NUMNP,NDOF,TIME,DT1,DT2,PRTC,PLTC,FRCI,PRTO,
      . PLTO,FRCO,VT,VR,AT,AR,UT,UR,XMST,XMSR,IRBODY,RBDYN,USRHV,
      . MESSAG,TOTALM,CYCL,IDRINT)
C*************************************************************
C |   LIVERMORE SOFTWARE TECHNOLOGY CORPORATION (LSTC)          |
C |   ------------------------------------------------------   |
C |   COPYRIGHT 1987, 1988, 1989 JOHN O. HALLQUIST, LSTC       |
C |   ALL RIGHTS RESERVED                                      |
C*************************************************************
C
      CHARACTER*(*) MESSAG
      INTEGER CYCLE
C
C
C     USER SUBROUTINE FOR SOLUTION CONTROL
C
C     NOTE:  LS-DYNA3D USED AN INTERNAL NUMBERING SYSTEM TO
C            ACCOMODATE ARBITRARY NODE NUMBERING.  TO ACCESS
C            INFORMATION FOR USER NODE N, ADDRESS ARRAY LOCATION M,
C            M=LQF(N,1).  TO OBTAIN USER NODE NUMBER, N,
C            CORRESPONDING TO ARRAY ADDRESS M, SET N=LQFINV(M,1)
C
C     ARGUMENTS:
C        NUMNP=NUMBER OF NODAL POINTS
C        NDOF=NUMBER OF DEGREES IF FREEDOM PER NODE
C        TIME=CURRENT SOLUTION TIME
C        PRTC=OUTPUT INTERVAL FOR TAURUS TIME HISTORY DATA
C        PLTC=OUTPUT INTERVAL FOR TAURUS STATE DATA
C        FRCI=OUTPUT INTERVAL FOR TAURUS INTERFACE FORCE DATA
C        PRTO=OUTPUT TIME FOR TIME HISTORY FILE
C        PLTO=OUTPUT TIME FOR STATE DATA
C        FRCO=OUTPUT TIME FOR FORCE DATA
C        VT(3,NUMNP)  =NODAL TRANSLATIONAL VELOCITY VECTOR
C        VR(3,NUMNP)  =NODAL ROTATIONAL VELOCITY VECTOR.  THIS ARRAY
C                      IS DEFINED IF AND ONLY IF NDOF=6
C        AT(3,NUMNP)  =NODAL TRANSLATIONAL ACCELERATION VECTOR
C        AR(3,NUMNP)  =NODAL ROTATIONAL ACCELERATION VECTOR.  THIS
C                      ARRAY IS DEFINED IF AND ONLY IF NDOF=6
C        UT(3,NUMNP)  =NODAL TRANSLATIONAL DISPLACEMENT VECTOR
C        UR(3,NUMNP)  =NODAL ROTATIONAL DISPLACEMENT VECTOR.  THIS ARRAY
C                      IS DEFINED IF AND ONLY IF NDOF=6
C        XMST(NUMNP)  =RECIPROCAL OF NODAL TRANSLATIONAL MASSES
C        XMSR(NUMNP)  =RECIPROCAL OF NODAL ROTATIONAL MASSES.  THIS
C                      ARRAY IS DEFINED IF AND ONLY IF NDOF=6
C        IRBODY       =FLAG FOR RIGID BODY NODAL POINTS
C                      IF DEFORMABLE NODE THEN SET TO 1.0
C                      IF RIGID BODY NODE THEN SET TO 0.0
C                      DEFINED IF AN ONLY IF RIGID BODY ARE PRESENT
```

```
C                      I.E.,IRBODY.NE.0 IF NO RIGID BODY ARE PRESENT
C        USRHV(LENHV)  =USER DEFINED HISTORY VARIABLES THAT ARE STORED
C                       IN THE RESTART FILE.  LENHV=100+U*NUMMAT WHERE
C                       NUMMAT IS THE # OF MATERIALS IN THE PROBLEM.
C                       ARRAY USRHV IS UPDATED ONLY IN THIS SUBROUTINE.
C        MESSAG        =FLAG FOR DYNA3D WHICH MAY BE SET TO:
C                       'SW1.' LS-DYNA3D TERMINATES WITH RESTART FILE
C                       'SW3.' LS-DYNA3D WRITES A RESTART FILE
C                       'SW4.' LS-DYNA3D WRITES A PLOT STATE
C        TOTALM        =TOTAL MASS IN PROBLEM
C           CYCLE =CYCLE NUMBER
C           IDRINT      =FLAG FOR DYNAMIC RELAXATION PHASE
C                        .NE.0:  DYNAMIC RELAXATION IN PROGRESS
C                        .EQ.0:  SOLUTION PHASE
C
      COMMON/PTIMES/  PRTIMS(32),PRTLST(32),IGMPRT
C
C     PRTIMS(32)=OUTPUT INTERVALS FOR ASCII FILES
C
C     ASCII FILES
C           ( 1)=CROSS SECTION FORCES
C           ( 2)=RIGID WALL FORCES
C           ( 3)=NODAL DATA
C           ( 4)=ELEMENT DATA
C           ( 5)=GLOBAL DATA
C           ( 6)=DISCRETE ELEMENTS
C           ( 7)=MATERIAL ENERGIES
C           ( 8)=NODAL INTERFACE FORCES
C           ( 9)=RESULTANT INTERFACE FORCES
C           (10)=SMUG ANIMATOR
C           (11)=SPC REACTION FORCES
C           (12)=NODAL CONSTRAIN RESULTANT FORCES
C           (13)=AIRBAG STATISTICS
C           (14)=AVS DATABASE
C           (15)=NODAL FORCE GROUPS
C           (16)=OUTPUT INTERVALS FOR NODAL BOUNDARY CONDITIONS
C           (17)-(32)=UNUSED AT THIS TIME
C
C     PRTLST(32)=OUTPUT TIMES FOR ASCII FILES ABOVE.  WHEN SOLUTION TIME
C             EXCEEDS THE OUTPUT TIME A PRINT STATE IS DUMPED.
C
      COMMON/RBKENG/ENRBDY,RBDYX,RBDYY,RBDYZ
C
C     TOTAL RIGID BODY ENERGIES AND MOMENTUMS:
C             ENRBDY=RIGID BODY KINETIC ENERGY
C             RBDYX =RIGID BODY X-MOMENTUM
C             RBDYY =RIGID BODY Y-MOMENTUM
C             RBDYZ =RIGID BODY Z-MOMENTUM
C
      COMMON/RBKENG/ENRBDY,RBDYX,RBDYY,RBDYZ
C
C     TOTAL RIGID BODY ENERGIES AND MOMENTUMS:
C             SWXMOM=STONEWALL X-MOMENTUM
C             SWYMOM=STONEWALL Y-MOMENTUM
C             SWZMOM=STONEWALL Z-MOMENTUM
C             ENRBDY=STONEWALL KINETIC ENERGY
C
      COMMON/DEENGS/DEENG
C
C     DEENG=TOTAL DISCRETE ELEMENT ENERGY
```

```
C
      COMMON/ENERGY/XPE
C
C     XPE   =TOTAL INTERNAL ENERGY IN THE FINITE ELEMENTS
C
      DIMENSION VT(3,*),VR(3,*),AT(3,*),AR(3,*),UT(3,*),UR(3,*)
      XMST(*),XMSR(*),RBDYN(*),USRHV(*)
C
C     SAMPLE MOMENTUM AND KINETIC ENERGY CALCULATIONS
C
C     REMOVE ALL COMMENTS IN COLUMN 1 BELOW TO ACTIVATE
CC
CC
CC    INITIALIZE KINETIC ENERGY, XKE, AND X,Y,Z MOMENTUMS.
CC
C     XKE=2.*SWKENG+2.*ENRBDY
C     XM-SWXMOM+RBDYX
C     YM=SWYMOM+RBDYY
C     ZM=SWZMOM+RBDYZ
CC
C     NUMNP2=NUMNP
C     IF (NDOF.EQ.6) THEN
C     NUMNP2=NUMNP+NUMNP
C     ENDIF
C     PRINT *,NDOF
C     IF(IRBODY.EQ.0) THEN
CC
CC
CC    NO RIGID BODIES PRESENT
CC
CC    NOTE IN BLANK COMMENT VR FOLLOWS VT.  THIS FACT IS USED BELOW.
C     DO 10 N=1,NUMNP2
C     XMSN=1./XMST(N)
C     VN1=VT(1,N)
C     VN2=VT(2,N)
C     VN3=VT(3,N)
C     XM=XM+XMSN*VN1
C     YM=YM+XMSN*VN2
C     ZM=ZM+XMSN*VN3
C     XKE=XKE+XMSN*(VN1*VN1+VN2*VN2+VN3*VN3)
C  10 CONTINUE
CC
C     ELSE
CC
CC    RIGID BODIES PRESENT
CC
C     DO 20 N=1,NUMNP
C     XMSN=1./XMST(N)
C     VN1=RBDYN(N)*VT(1,N)
C     VN2=RBDYN(N)*VT(2,N)
C     VN3=RBDYN(N)*VT(3,N)
C     XM=XM+XMSN*VN1
C     YM=YM+XMSN*VN2
C     ZM=ZM+XMSN*VN3
C     XKE=XKE+XMSN*(VN1*VN1+VN2*VN2+VN3*VN3)
C  20 CONTINUE
C     IF (NDOF.EQ.6) THEN
C     DO 30 N=1,NUMNP
C     XMSN=1./XMSR(N)
C     VN1=RBDYN(N)*VR(1,N)
```

```
C     VN2=RBDYN(N)*VR(2,N)
C     VN3=RBDYN(N)*VR(3,N)
C     XM=XM+XMSN*VN1
C     YM=YM+XMSN*VN2
C     ZM=ZM+XMSN*VN3
C     XKE=XKE+XMSN*(VN1*VN1+VN2*VN2+VN3*VN3)
C  30 CONTINUE
C     ENDIF
CC
C     ENDIF
      RETURN
      END
CC
CC.....TOTAL KINETIC ENERGY
C      XKE=.5*XKE
CC.....TOTAL INTERNAL ENERGY
C      XIE=.XPE+DEENG
CC.....TOTAL ENERGY
C      XTE=XKE+XPE+DEENG
CC.....TOTAL X-RIGID BODY VELOCITY
C      XRBV=XM/TOTALM
CC.....TOTAL Y-RIGID BODY VELOCITY
C      YRBV=YM/TOTALM
CC.....TOTAL Z-RIGID BODY VELOCITY
C      ZRBV=ZM/TOTALM
C
      RETURN
      END
```

APPENDIX D: User Defined Interface Control

This subroutine may be provided by the user to turn the interfaces on and off. This option is activated by the *USER_INTERFACE_CONTROL keyword. The arguments are defined in the listing provided below.

```
      SUBROUTINE UCTRL2 (NSI,NTY,TIME,CYCLE,MSR,NMN,NSV,NSN,
     1     THMR,THSV,VT,XI,UT,ISKIP,IDRINT,NUMNP,DT2,NINPUT,UA)
C*************************************************************************
C |   LIVERMORE SOFTWARE TECHNOLOGY CORPORATION (LSTC)                     |
C |   ---------------------------------------------------------------     |
C |   COPYRIGHT 1987, 1988, 1989 JOHN O. HALLQUIST, LSTC                   |
C |   ALL RIGHTS RESERVED                                                  |
C*************************************************************************
C
      INTEGER CYCLE
C
C
C     USER SUBROUTINE FOR INTERFACE CONTROL
C
C     NOTE:  LS-DYNA3D USED AN INTERNAL NUMBERING SYSTEM TO
C            ACCOMODATE ARBITRARY NODE NUMBERING.  TO ACCESS
C            INFORMATION FOR USER NODE N, ADDRESS ARRAY LOCATION M,
C            M=LQF(N,1).  TO OBTAIN USER NODE NUMBER, N,
C            CORRESPONDING TO ARRAY ADDRESS M, SET N=LQFINV(M,1)
C
C     ARGUMENTS:
C        NSI          =NUMBER OF SLIDING INTERFACE
C        NTY          =INTERFACE TYPE.
C                      .EQ.4:SINGLE SURFACE
C                      .NE.4:SURFACE TO SURFACE
C        TIME         =CURRENT SOLUTION TIME
C        CYCLE        =CYCLE NUMBER
C        MSR(NMN)     =LIST OF MASTER NODES NUMBERS IN INTERNAL
C                       NUMBERING SCHEME
C        NMN          =NUMBER OF MASTER NODES
C        NSV(NSN)     =LIST OF SLAVE NODES NUMBERS IN INTERNAL
C                       NUMBERING SCHEME
C        NSN          =NUMBER OF SLAVE NODES
C        THMR(NMN)    =MASTER NODE THICKNESS
C        THSV(NSN)    =SLAVE NODE THICKNESS
C        VT(3,NUMNP)  =NODAL TRANSLATIONAL VELOCITY VECTOR
C        XI(3,NUMNP)  =INITIAL COORDINATES AT TIME=0
C        UT(3,NUMNP)  =NODAL TRANSLATIONAL DISPLACEMENT VECTOR
C        IDRINT       =FLAG FOR DYNAMIC RELAXATION PHASE
C                      .NE.0:DYNAMIC RELAXATION IN PROGRESS
C                      .EQ.0:SOLUTION PHASE
C        NUMNP        =NUMBER OF NODAL POINTS
C        DT2          =TIME STEP SIZE AT N+1/2
C        NINPUT       =NUMBER OF VARIABLES INPUT INTO UA
C        UA(*)        =USER'S ARRAY, FIRST NINPUT LOCATIONS
C                       DEFINED BY USER.  THE LENGTH OF THIS
C                       ARRAY IS DEFINED ON CONTROL CARD 10.
C                       THIS ARRAY IS UNIQUE TO INTERFACE NSI.
C
C     SET FLAG FOR ACTIVE CONTACT
```

```
C       ISKIP=0 ACTIVE
C       ISKIP=1 INACTIVE
C
C***********************************************************************
      DIMENSION MSR(*),NSV(*),THMR(*),THSV(*),VT(3,*),XI(3,*),
                UT(3,*)UA(*)
C
C     THE FOLLOWING SAMPLE OF CODEING IS PROVIDED TO ILLUSTRATE HOW
C     THIS SUBROUTINE MIGHT BE USED.  HERE WE CHECK TO SEE IF THE
C     SURFACES IN THE SURFACE TO SURFACE CONTACT ARE SEPARATED.  IF
C     SO THE ISKIP=1 AND THE CONTACT TREATMENT IS SKIPPED.
C
      IF (NTY.EQ.4) RETURN
      DT2HLF=DT2/2.
      XMINS= 1.E20
      XMAXS=-XMINS
      YMINS= 1.E20
      YMAXS=-YMINS
      ZMINS= 1.E20
      ZMAXS=-ZMINS
      XMINM= 1.E20
      XMAXM=-XMINM
      YMINM= 1.E20
      YMAXM=-YMINM
      ZMINM= 1.E20
      ZMAXM=-ZMINM
      THKS=0.0
      THKM=0.0
      DO 10 I=1,NSN
      DSP1=UT(1,NSV(I))+DT2HLF*VT(1,NSV(I))
      DSP2=UT(2,NSV(I))+DT2HLF*VT(2,NSV(I))
      DSP3=UT(3,NSV(I))+DT2HLF*VT(3,NSV(I))
      X1=XI(1,NSV(I))+DSP1
      X2=XI(2,NSV(I))+DSP2
      X3=XI(3,NSV(I))+DSP3
      THKS =MAX(THSV(I),THKS)
      XMINS=MIN(XMINS,X1)
      XMAXS=MAX(XMAXS,X1)
      YMINS=MIN(YMINS,X2)
      YMAXS=MAX(YMAXS,X2)
      ZMINS=MIN(ZMINS,X3)
      ZMAXS=MAX(ZMAXS,X3)
   10 CONTINUE
      DO 20 I=1,NMN
      DSP1=UT(1,MSR(I))+DT2HLF*VT(1,MSR(I))
      DSP2=UT(2,MSR(I))+DT2HLF*VT(2,MSR(I))
      DSP3=UT(3,MSR(I))+DT2HLF*VT(3,MSR(I))
      X1=XI(1,MSR(I))+DSP1
      X2=XI(2,MSR(I))+DSP2
      X3=XI(3,MSR(I))+DSP3
      THKM =MAX(THMR(I),THKS)
      XMINS=MIN(XMINM,X1)
      XMAXS=MAX(XMAXM,X1)
      YMINS=MIN(YMINM,X2)
      YMAXS=MAX(YMAXM,X2)
      ZMINS=MIN(ZMINM,X3)
      ZMAXS=MAX(ZMAXM,X3)
   20 CONTINUE
      IF (XMAXS+THKS.LT.XMINM-THKM) GO TO 40
      IF (YMAXS+THKS.LT.YMINM-THKM) GO TO 40
```

```
      IF (ZMAXS+THKS.LT.ZMINM-THKM) GO TO 40
      IF (XMAXS+THKM.LT.XMINS-THKS) GO TO 40
      IF (YMAXS+THKM.LT.YMINS-THKS) GO TO 40
      IF (ZMAXS+THKM.LT.ZMINS-THKS) GO TO 40
      ISKIP=0
      RETURN
   40 ISKIP=1
      RETURN
      END
```

APPENDIX E: User Defined Interface Friction

This subroutine may be provided by the user to set the Coulomb friction coefficients. This option is activated by the *USER_INTERFACE_FRICTION keyword. The arguments are defined in the listing provided below.

```
      SUBROUTINE USRFRC (NSI,TIME,CYCLE,DT2,NSLAVE,AREAS,XS,YS,ZS,
     . MSN,MASTRS,AREAM,XCM,YCM,ZCM,STFSN,STFMS,FORCEN,RVX,RVY,RVZ,
     . FRIC1,FRIC2,FRIC3,FRIC4,NINPUT,UA,SIDE)
C*************************************************************
C |   LIVERMORE SOFTWARE TECHNOLOGY CORPORATION (LSTC)          |
C |   ------------------------------------------------------   |
C |   COPYRIGHT 1987, 1988, 1989 JOHN O. HALLQUIST, LSTC       |
C |   ALL RIGHTS RESERVED                                      |
C*************************************************************
C
      INTEGER CYCLE
      CHARACTER*(*) SIDE
      DIMENSION UA(*),MASTRS(4),XCM(4),YCM(4),ZCM(4)
C
C
C     USER SUBROUTINE FOR INTERFACE FRICTION CONTROL
C
C     NOTE:  LS-DYNA3D USES AN INTERNAL NUMBERING SYSTEM TO
C            ACCOMODATE ARBITRARY NODE NUMBERING.  TO ACCESS
C            INFORMATION FOR USER NODE N, ADDRESS ARRAY LOCATION M,
C            M=LQF(N,1).  TO OBTAIN USER NODE NUMBER, N,
C            CORRESPONDING TO ARRAY ADDRESS M, SET N=LQFINV(M,1)
C
C     ARGUMENTS:
C        NSI          =NUMBER OF SLIDING INTERFACE
C        TIME         =CURRENT SOLUTION TIME
C        CYCLE        =CYCLE NUMBER
C        DT2          =TIME STEPS SIZE AT N+1/2
C        NSLAVE       =SLAVE NODE NUMBER IN LS-DYNA3D INTERNAL
C                      NUMBERING
C        AREAS        =SLAVE NODE AREA (INTERFACE TYPES 5&10 ONLY)
C        XS           =X-COORDINATE SLAVE NODE (PROJECTED)
C        YS           =Y-COORDINATE SLAVE NODE (PROJECTED)
C        ZS           =Z-COORDINATE SLAVE NODE (PROJECTED)
C        MSN          =MASTER SEGMENT NUMBER
C        MASTRS(4)    =MASTER SEGMENT NODE IN LS-DYNA3D INTERNAL
C                      NUMBERING
C        AREAM        =MASTER SEGMENT NUMBER
C        XCM(4)       =X-COORDINATES MASTER SURFACE (PROJECTED)
C        YCM(4)       =Y-COORDINATES MASTER SURFACE (PROJECTED)
C        ZCM(4)       =Z-COORDINATES MASTER SURFACE (PROJECTED)
C        STFSN        =SLAVE NODE PENALTY STIFFNESS
C        STFMS        =MASTER SEGMENT PENALTY STIFFNESS
C        FORCEN       =NORMAL FORCE
C        RVX,RVY,RVZ,=RELATIVE X,Y,Z-VELOCITY BETWEEN SLAVE NODE AND
                      MASTER SEGMENT
```

```
C*************************************************************
C     THE FOLLOWING VALUES ARE TO BE SET BY USER
C
C        FRIC1        =STATIC FRICTION COEFFICIENT
C        FRIC2        =DYNAMIC FRICTION COEFFICIENT
C        FRIC3        =DECAY CONSTANT
C        FRIC4        =VISCOUS FRICTION COEFFICIENT (SETTING FRIC4=0
                       TURNS THIS OPTION OFF)
C
C*************************************************************
C
C        NINPUT       =NUMBER OF VARIABLES INPUT INTO UA
C        UA(*)        =USERS' ARRAY, FIRST NINPUT LOCATIONS
C                      DEFINED BY USER.  THE LENGTH OF THIS
C                      ARRAY IS DEFINED ON CONTROL CARD 15.
C                      THIS ARRAY IS UNIQUE TO INTERFACE NSI.
C
C        SIDE         ='MASTER' FOR FIRST PASS.  THE MASTER
C                      SURFACE IS THE SURFACE DESIGNATED IN THE
C                      INPUT.
C                     ='SLAVE' FOR SECOND PASS AFTER SLAVE AND
C                      MASTER SURFACES HAVE BE SWITCHED FOR
C                      THE TYPE 3 SYMMETRIC INTERFACE TREATMENT
C
C*************************************************************
C
      RETURN
      END
```

APPENDIX F: Occupant Simulation Including the Coupling to Programs CAL3D and MADYMO

INTRODUCTION

LS-DYNA3D is coupled to occupant simulation codes to generate solutions in automotive crashworthiness that include occupants interacting with the automotive structure. In such applications LS-DYNA3D provides the simulation of the structural and deformable aspects of the model and the OSP (Occupant Simulation Program) simulates the motion of the occupant. There is some overlap between the two programs which provides flexibility in the modeling approach. For example, both the OSP and LS-DYNA3D have the capability of modeling seat belts and other deformable restraints. The advantage of using the OSP is related to the considerable databases and expertise that have been developed in the past for simulating dummy behavior using these programs.

The development of the interface provided LSTC a number of possible approaches. The approach selected is consistent with the LSTC philosophy of providing the most flexible and useful interface possible. This is important because the field of non-linear mechanics is evolving rapidly and techniques which are used today are frequently rendered obsolete by improved methodologies and lower cost computing which allows more rigorous techniques to be used. This does make the learning somewhat more difficult as there is not any single procedure for performing a coupling.

One characteristic of LS-DYNA3D is the large number of capabilities, particularly those associated with rigid bodies. This creates both an opportunity and a difficulty: LSDYNA3D has many ways approximating different aspects of problems, but they are frequently not obvious to users without considerable experience. Therefore, in this Appendix we emphasize modeling methods rather than simply listing capabilities.

THE LS-DYNA3D/OCCUPANT SIMULATION PROGRAM LINK

Coupling between the OSP and LS-DYNA3D is performed by combining the programs into a single executable. In the case of CAL3D, LS-DYNA3D calls CAL3D as a subroutine, but in the case of MADYMO, LS-DYNA3D is called as a subroutine. The two programs are then integrated in parallel with the results being passed between the two until a user defined termination time is reached.

The OSP and LS-DYNA3D have different approaches to the time integration schemes. The OSP time integrators are based on accurate implicit integrators which are valid for large time steps which are on the order of a millisecond for the particular applications of interest here. An iterative solution is used to insure that the problem remains in equilibrium. The implicit integrators are

extremely good for smoothly varying loads, however, sharp nonlinear pulses can introduce considerable error. An automatic time step size control which decreases the time step size quickly restores the accuracy for such events. The LS-DYNA3D time integrator is based on an explicit central difference scheme. Stability requires that the time step size be less than the highest frequency in the system. For a coarse airbag mesh, this number is on the order of 100 microseconds while an actual car crash simulation is on the order of 1 microsecond. The smallest LS-DYNA3D models have at least 1,000 elements. Experience indicates that the cost of a single LS-DYNA3D time step for a small model is at least as great as the cost of a time step in the OSP. Therefore, in the coupling, the LS-DYNA3D time step is used to control the entire simulation including the OSP part. This approach has negligible cost penalties and avoids questions of stability and accuracy that would result by using a subcycling scheme between the two programs. Optionally, a subcycling scheme can be used, however, the results of the analysis have to be checked with care.

LS-DYNA3D has a highly developed rigid body capability which is used in different parts of automobile crash simulation. In particular, components such as the engine are routinely modeled with rigid bodies. These rigid bodies have been modified so that they form the basis of the coupling procedure in LS-DYNA3D to the OSP.

In LS-DYNA3D, the geometry of a model is broken down into nodal points which identify positions in space. These nodes are then connected by elements so that the volume of a structure is identified. Each element has a “material” associated with it. If the element is deformable, then the material will specify its characteristics such as density and Young’s Modulus. A crash model can consist of 100 or more separate materials which are each assigned a “material number,” and each material number has an associated “material type” which determines if it is elastic, plastic, viscoelastic, orthotropic, etc.

The material type may also specify that it is a rigid body. In this case, all elements of the same material number are treated as a single rigid body. These elements are integrated to determine the mass, centroid and moments of inertia for the group. This group is then treated as a rigid body with six degrees-of-freedom including three translations and three rotations. The positions of the rigid bodies are updated in LS-DYNA3D by a time integrator which works together with the central difference time integration.

There is an additional flag which specifies that the LS-DYNA3D rigid body is coupled to an OSP rigid body. This flag can be found in the description of the rigid body material *MAT_RIGID (formerly material type 20). In coupled updates, the OSP rigid body time integrator takes over control of the LS-DYNA3D rigid body and the normal LS-DYNA3D updates are bypassed. The time integration procedure is then as follows:

1. At the beginning of a step, LS-DYNA3D determines the locations and updates the positions of all of the rigid bodies which are coupled to the OSP. This information is obtained from common block information in the OSP.

2. Using the information on rigid body locations, LS-DYNA3D proceeds to update the stresses and history variables of all of the deformable structures and computes the resultant forces acting on all rigid bodies.

3. The resultant forces are stored into an OSP common block along with the current time step. Control is then returned to the OSP so that the step can be completed by the OSP determining the new positions of the rigid bodies based on the applied forces.

At the end of the calculation LS-DYNA3D terminates normally, closing its files, and then control is returned to OSP which will also terminate normally. The termination time for the coupled run is taken as the minimum of the termination time provided to LS-DYNA3D and the termination time provided to the OSP.

The executable for the coupling with MADYMO currently needs to be specially created at each site. TNO provides all of the appropriate load modules with their libraries, and the appropriate load modules for LS-DYNA3D may be obtained by the corporate contact point at the LS-DYNA3D distributor. A complete executable must then be made by linking the two libraries. A revised password file must be obtained from TNO prior to running the coupled code. Coupling with CAL3D requires special on-site modification of the client's CAL3D version to eliminate conflicting I/O unit numbers and to ensure that the common block lengths between the codes are consistent. LSTC does not distribute or support CAL3D.

To make the coupled program run, an input deck must be provided to both the OSP and LS-DYNA3D. The two input decks must be provided in the same set of consistent units. This can potentially require a major conversion to either the OSP input or the LS-DYNA3D input. With two legitimate and consistent input decks, the coupled program should run to completion with no problems. Additional inputs are required to make the models interact between the OSP and LS-DYNA3D portions of the run.

The simplest form of a coupled simulation is simply to include a single body in an OSP run. No special modifications are needed to the OSP input deck for use in the coupled simulation. Ellipsoids and planes in the OSP are usually attached to "segments" which correspond to LS-DYNA3D "rigid bodies." Because the coupling procedure works on the basis of shared information on LS-DYNA3D rigid bodies with the OSP segments, the ellipsoids/planes listed in the OSP section must correspond to the segments which are to be coupled. These ellipsoids and planes

may be actual geometry which is used for contact, or they may be simply artificial shapes to permit the data transfer between the OSP and LS-DYNA3D.

DUMMY MODELING

The dummy is typically modeled entirely within the OSP. The coupling of the dummy into LS-DYNA3D requires the creation of a separate LS-DYNA3D rigid body material for each segment of the OSP. The easiest way to create a mesh for the model is to set the LS-DYNA3D rigid body coupling option to 2.0. This caused LS-DYNA3D to search all of the ellipsoids connected to the appropriate segment and generate meshes which are then slaved the OSP dummy. Thus, with minimal input, a complete dummy may be generated and the kinematics may be traced in LS-DYNA3D and displayed in the LS-DYNA3D post-processor, LS-TAURUS

Once the basic dummy coupling has been accomplished, the deformable finite element structure can be added. Assuming that an ellipsoid is available for the steering wheel, a flat airbag can be added in the proper location. One or more nodes must be attached to the steering wheel. This is done by identifying the attached nodes as “Extra Nodes for Rigid Body” which is input in LS-DYNA3D by *CONSTRAINED_EXTRA_NODES_Option. The nodes are slaved to the LS-DYNA3D material which has been coupled to the MADYMO steering wheel model. Contact must now be identified between the airbag and the steering wheel, the windshield, and the various body parts which may be affected. This requires the use of one geometric contact entity (see *CONTACT_ENTITY) for each plane or ellipsoid which may interact with the airbag. A control volume specifying inflation properties for the airbag must be specified (see *AIRBAG_OPTION) to complete the model.

AIRBAG MODELING

Modeling of airbags is accomplished by use of shell or membrane elements in conjunction with a control volume (see *AIRBAG_OPTION) and possibly a single surface contact algorithm to eliminate interpenetrations during the inflation phase (see *CONTACT_OPTION). The contact types showing an “a” in front are most suited for airbag analysis. Current recommended material types for the airbags are:

*MAT_ELASTIC = Type 1.	Elastic
*MAT_COMPOSITE_DAMAGE = Type 22.	Layered orthotropic elastic for composites
*MAT_FABRIC = Type 34.	Fabric model for folded airbags

Model 34 is a "fabric" model which can be used for flat bags. As a user option this model may or may not support compression.

The elements which can be used are as follows:

Belytschko-Tsay quadrilateral with 1 point quadrature. This element behaves rather well for folded and unfolded cases with only a small tendency to hourglass. The element tends to be a little stiff. Stiffness form hourglass control is recommended.

Belytschko-Tsay membrane. This model is softer than the normal Belytschko-Tsay element and can hourglass quite badly. Stiffness form hourglass is recommended. As a better option, the fully integrated Belytschko-Tsay membrane element can be chosen.

C0 Triangular element. The C0 triangle is very good for flat bag inflation and has no tendency to hourglass.

The best choice is a specially developed airbag membrane element with quadrilateral shape. This is an automatic choice when the fabric material is used.

As an airbag inflates, a considerable amount of energy is transferred to the surrounding air. This energy transfer decreases the kinetic energy of the bag as it inflates. In the control volume logic, this is simulated either by using either a mass weighted damping option or a back pressure on the bag based on a stagnation pressure. In both cases, the energy that is absorbed is a function of the fabric velocity relative to a rigid body velocity for the bag. For the mass weighted case, the damping force on a node is proportional to the mass times the damping factor times the velocity vector. This is quite effective in maintaining a stable system, but has little physical justification. The latter approach using the stagnation pressure method estimates the pressure needed to accelerate the surrounding air to the speed of the fabric. The formula for this is:

$$P = Area \times \alpha \times \left(\left(\vec{V}_i - \vec{V}_{cg} \right) \cdot \hat{n} \right)^2$$

This formula accomplishes a similar function and has a physical justification. Values of the damping factor, α, are limited to the range of 0 to 1, but a value of 0.1 or less is more likely to be a good value.

KNEE BOLSTER

The knee-to-knee bolster interactions are characterized by the stiffness of the knee being comparable to that of the knee bolster. Therefore, modeling the knee as a rigid body may produce large errors in the interaction forces. Calibrated force-deflection curves could be determined, but they would have no predictive value for slight changes to knee bolster designs. For this reason, a more accurate modeling of the compliance of the knee bolster and the knee is required.

The knee can be modeled as a combined rigid/deformable body. The rigid body is coupled to the OSP. Overlaying the rigid body are brick elements which model the "skin" that exists over the knees of the dummy. These brick elements use material type 6 (*MAT_VISCOELASTIC) which is a viscoelastic model that does a reasonable job of approximating the hysteretic behavior of rubbers. The inner layer of the brick elements is attached to the rigid body through the *CONSTRAINED_EXTRA_NODES Option. Between the knee bolster is a SURFACE-TO-SURFACE contact definition.

COMMON ERRORS

1. Improper airbag inflation or no inflation.

The most common problem is inconsistency in the units used for the input constants. An inflation load curve must also be specified. The normals for the airbag segments must all be consistent and facing outwards. If a negative volume results, this can sometimes be quickly cured by using the "flip" flag on the control volume definition to force inward facing normals to face outwards.

2. Excessive airbag distortions.

Check the material constants. Triangular elements should have less distortion problems than quadrilaterals. Overlapped elements at time zero can cause locking to occur in the contact leading to excessive distortions. The considerable energy input to the bag will create numerical noise and some damping is recommended to avoid problems.

3. The dummy passes through the airbag.

A most likely problem is that the contacts are improperly defined. Another possibility is that the models were developed in an incompatible unit system. The extra check for penetration flag if set to 1 on the contact control cards variable PENCHK in the *CONTACT_... definitions may sometimes cause nodes to be prematurely released due to the softness of the penalties. In this case the flag should be turned off.

4. The OSP fails to converge.

This may occur when excessively large forces are passed to the OSP. First, check that unit systems are consistent and then look for improperly defined contacts in the LS-DYNA3D input.

5. Time step approaches zero.

This is almost always in the airbag. If elastic or orthotropic (*MAT_ELASTIC or *MAT_COMPOSITE material 1 or 22) is being used, then switch to fabric material *MAT_FABRIC which is less time step size sensitive and use the fully integrated membrane element. Increasing the damping in the control volume usually helps considerably. Also, check for “cuts” in the airbag where nodes are not merged. These can allow elements to deform freely and cut the time step to zero.

APPENDIX G: Interactive Graphics Commands

Only the first four or less characterers of command are significant. These commands are available in the interactive phase of LS-DYNA3D. The interactive graphics are available by using the "SW5." command after invoking the Ctrl-C interrupt. The MENU command brings up a push button menu.

ANIMATE	Animate saved sequence, stop with switch 1.
BACK	Return to previous display size after zoom, then list display attributes.
BGC	Change display background color RGB proportions BGC <red> <green> <blue>.
BIP	Select beam integration point for contour; BIP <#>.
CENTER	Center model, center on node, or center with mouse, i.e., center cent <value> or cent gin.
CL	Classification labels on display; class commercial_in_confidence.
CMA	Color materials on limited color displays.
COLOR	Set or unset shaded coloring of materials.
CONTOUR	View with colored contour lines; contour <component #> <list mat #>; see TAURUS manual.
COOR	Get node information with mouse.
COP	Hardcopy of display on the PC copy <laserj paintj tekcol coljet or epson>.
CR	Restores cutting plane to default position.
CUT	Cut away model outside of zoom window; use mouse to set zoom window size.

CX	Rotate slice plane at zmin about x axis.
CY	Rotate slice plane at zmin about y axis.
CZ	Rotate slice plane at zmin about z axis.
DIF	Change diffused light level for material; DIF <mat #, -1 for all> <value>.
DISTANCE	Set distance of model from viewer; DIST <value in normalized model dimensions>.
DMATERIALS	Delete display of material in subsequent views; DMAT <ALL or list of numbers>.
DRAW	Display outside edges of model.
DSCALE	Scale current displacement from initial shape.
DYN	After using TAURUS command will reset display to read current DYNA3D state data.
ELPLT	Set or unset element numbering in subsequent views.
END	Delete display and return to execution.
ESCAPE	Escapes from menu pad mode.
EXECUTE	Return to execution and keep display active.
FCL	Fix or unfix current contour levels.
FOV	Set display field of view angle; FOV <value in degrees>.
FRINGE	View with colored contour fringes; fringe <component #> <list mat #>; see TAURUS manual.
GETFRAME	Display a saved frame; GETF <frame #>.

HARDWARE	Hardware mode; workstation hardware calls are used to draw, move and color model; repeat command to reset to normal mode.
HELP	
HZB	Switch on or off hardware zbuffer for a subsequent view, draw or contour command; rotations and translations will be in hardware.
LIMIT	Set range of node numbers subsequent views; limit <first node #> <last node #>.
MAT	Re-enable display of deleted materials mat <all or list of numbers>.
MENU	Button menu pad mode.
MOTION	Motion of model through mouse movement or use of a dial box. The left button down enables translation in the plane, middle button rotation about axes in the plane; and with right button down in the out of plane axis; left and middle button down quit this mode.
MOV	Drag picked part to new position set with mouse.
NDPLT	Set or unset node numbering in subsequent views.
NOFRAME	Set and unset drawing of a frame around the picture.
PAUSE	Animation display pause in seconds
PHS2 or THISTORY	Time history plotting phase. Similar to LS-TAURUS.
PICK	Get element information with mouse.
POST	Enable or disenable postscript mode on the PC and eps file is written as picture is drawn; remove eofs and initgraphics for eps use.

QUIT	Same as execute.
RANGE	Set fix range for contour levels; range <minvalue> <maxvalue>.
RAX	Reflect model about xy plane; restore command will switch-off reflections.
RAY	Reflect model about yz plane; restore command will switch-off reflections.
RAZ	Reflect model about zx plane, restore command will switch-off reflections.
RESTORE	Restores model to original position, also switches off element and node numbers, slice capper, reflections and cut model.
RETURN	Exit.
RGB	Change color red green blue element <mat #> <red> <green> <blue>.
RX	Rotate model about x axis.
RY	Rotate model about y axis.
RZ	Rotate model about z axis.
SAVE	Set or unset saving of display for animation.
SEQUENCE	Periodic plot during execution; SEQ <# of cycles> <commands> EXE.
SHR	Shrink element facets towards centoids in subsequent views, shrink <value>.
SIP	Select shell integration point for contour; SIP <#>.

SLICE	Slice model a z-minimum plane; slice <value in normalized model dimension> this feature is removed after using restore. Slice enables internal details for brick elements to be used to generate new polygons on the slice plane.
SNORMAL	Set or unset display of shell direction normals to indicate topology order.
SPOT	Draw node numbers on model spot <first #> <last # for range>.
TAURUS	LS-TAURUS database, TAU <state #>, or state <state #>, reads LS-TAURUS file to extract previous state data.
TRIAD	Set or unset display of axis triad.
TSHELL	Set or unset shell element thickness simulation in subsequent views.
TV	Change display type.
TX	Translates model along x axis.
TY	Translates model along y axis.
TZ	Translates model along z axis.
V	Display model using painters algorithm.
VECTOR v or d	View with vector arrows of velocity or displacement; <v> or <d>.
ZB	Switch on or off zbuffer algorithm for subsequent view; or draw commands.
ZIN	Zoom in using mouse to set display size and position.
ZMA	Set position of zmax plane; ZMAX <value in normalized model dimesions>.

ZMI	Set position of zmin plane; ZMIN <value in normalized model dimesions>.
ZOUT	Zoom out using mouse to set displays size expansion and position.

APPENDIX H: Interactive Material Model Driver

INTRODUCTION

The interactive material model driver in LS-DYNA3D allows calculation of the material constitutive response to a specified strain path. Since the constitutive model subroutines in LS-DYNA3D are directly called by this driver, the behavior of the constitutive model is precisely that which can be expected in actual applications. In the current implementation the constitutive subroutines for both shell elements and solid elements can be examined.

INPUT DEFINITION

The material model driver is invoked by setting the total number of beam, shell, and solid elements to zero in a standard LS-DYNA3D input file. The number of material model definitions should be set to one, the number of load curves should be nine, and the termination time to the desired length of the driver run. The complete state dump interval is interpreted as the time step to be used in the material model driver run. Plotting information is saved for every step of a driver run and sufficient memory is allocated to save this information in core for the interactive plotting phase.

The input deck consists only of the TITLE card, the CONTROL cards, one MATERIAL DEFINITION, and NINE LOAD CURVES describing the strain path should be defined. These nine curves define the time history of the displacement gradient components shown in Table H.1.

The velocity gradient matrix, L_{ij}, is approximated by taking the time derivative of the components in Table H.1. If these components are considered to form a tensor S_{ij} , then

$$L_{ij}(t) = \frac{S_{ij}(t) - S_{ij}\left(t_{k-1}\right)}{\left(t - t_k\right)}$$

and the strain rate tensor is defined as

$$d_{ij} = \frac{L_{ij} + L_{ij}^t}{2}$$

and the spin tensor as

$$\omega_{ij} = \frac{L_{ij} + L_{ij}^t}{2}$$

Table H.1. Load Curve Definitions versus Time

Load Curve Number	Component Definition
1	$\frac{\partial u}{\partial x}$
2	$\frac{\partial v}{\partial y}$
3	$\frac{\partial w}{\partial z}$
4	$\frac{\partial u}{\partial y}$
5	$\frac{\partial v}{\partial x}$
6	$\frac{\partial u}{\partial z}$
7	$\frac{\partial w}{\partial x}$
8	$\frac{\partial v}{\partial z}$
9	$\frac{\partial w}{\partial y}$

INTERACTIVE DRIVER COMMANDS

After reading the input file and completing the calculations, LS-DYNA3D gives a command prompt to the terminal. A summary of the available interactive commands is given below. An on-line help package is available by typing HELP.

ACCL	Scale all abscissa data by f. Default is f=1.
ASET amin omax	Set min and max values on abscissa to amin and amax, respectively. If amin=amax=0, scaling is automatic.
CHGL n	Change label for component n. LS-DYNA3D prompts for new label.
CONTINUE	Re-analyze material model.
CROSS c_1 c_2	Plot component c_1 versus c_2.
ECOMP	Display component numbers on the graphics display: 1 x-stress, 2 y-stress, 3 z-stress, 4 xy-stress, 5 yz-stress, 6 zx-stress, 7 effective plastic strain, 8 pressure, 9 von Mises (effective) stress, 10 1st principal deviatoric stress, 11 2nd principal deviatoric stress, 12 3rd principal deviatoric stress, 13 maximum shear stress, 14 1st principal stress, 15 2nd principal stress, 16 3rd principal stress, 17 ln (v / v0),

	18 relative volume, 19 v0 / v - 1.0, 20 1st history variable, 21 2nd history variable. Adding 100 or 400 to component numbers 1-16 yields strains and strain rates, respectively.
FILE name	Change pampers filename to name for printing.
GRID	Graphics displays will be overlaid by a grid of orthogonal lines.
NOGRID	Graphics displays will not be overlaid by a grid of orthogonal lines.
OSCL	Scale all ordinate data by f. Default is f=1.
OSET omin omax	Set min and max values on ordinate to omin and omax, respectively. If omin=omax=0, scaling is automatic.
PRINT	Print plotted time history data into file "pampers." Only data plotted after this command is printed. File name can be changed with the "file" command.
QUIT, END, T	Exit the material model driver program.
RDLC m n r_1 z_1 ... r_n z_n	Redefine load curve m using n coordinate pairs (r_1,z_1) (r_2,z_2),...(r_n,z_n).
TIME c	Plot component c versus time.
TV n	Use terminal output device type n. LS-DYNA3D provides a list of available devices.

Presently, the material model drive is implemented for solid and shell element material models. The driver does not yet support material models for beam elements.

APPENDIX I: VDA Database

VDA surfaces describe the surface of geometric entities and are useful for the simulation of sheet forming problems. The German automobile and automotive supplier industry (VDA) has defined the VDA guidelines [VDA, 1987] for a proper surface definition used for the exchange of surface data information. In LS-DYNA3D, this format can be read and used directly. Some files have to be provided for proper linkage to the motion of the correlation parts/materials in LS-DYNA3D.

Linking is performed via names. To these names surfaces are attached, which in turn can be linked together from many files externally to LS-DYNA3D. Thus, arbitrary surfaces can be provided by a preprocessor and then can be written to various files. The so called VDA file given on the LS-DYNA3D execution line via V=**vda** contains references to all other files. It also contains several other parameters affecting the treatment in the contact subroutines; see below.

The procedure is as follows. If VDA surfaces are to be used, the file specified by **vda** must have the following form. The file is free formatted with blanks as delimiters. Note that the characters "}" and "{" must be separated from the other input by spaces or new lines. The **vda** file may contain any number of input file specifications of the form:

file **afile bfile** {
    alias definitions
        }
    alias definitions
    followed by optional runtime parameters and a final end statement.

The file, **afile,** is optional, and if given must be the name of an ASCII input file formatted in accordance with the VDA Surface Interface Definitions as defined by the German automobile and automotive supply industry. **bfile** is required, and is the name of a binary VDA file. In a first run **afile** is given and **bfile** is created. In any further run, if the definitions have not changed, **afile** can be dropped and only **bfile** is needed. The purpose of **bfile** is that it allows for much faster initialization if the same VDA surfaces are to be used in a future LS-DYNA3D run.

If **afile** is given, **bfile** will always be created or overwritten. The alias definitions are used for linking to LS-DYNA3D and between the various surface definitions in the files defined by **afile** and **bfile.**

The alias definitions are of the form

alias **name** { el1 el2 ... eln }

where **name** is any string of up to 12 characters, and el1,...,eln are the names of VDA elements as specified in **afile**. The list of elements can be empty, in which case all the SURF and FACE VDA elements in **afile** will be used. Care should be taken to ensure that the alias **name** is unique, not only among the other aliases, but among the VDA element names in **afile**. This collection of VDA elements can later be indicated by the alias **name**. In particular, **name** may appear in later alias definitions.

Often it is required that a punch or die be created by a simple offset. This can be achieved in the **vda** files in two ways, either on VDA elements directly, or on parts defined by aliases. This feature offers great capability in generating and using surface data information.

Offset version 1:

As an option, the keyword **offset** may appear in the alias list which allows a new surface to be created as a normal offset (plus translation) of a VDA element in the file. The keyword **offset** my be applied to VDA elements only, not aliases. The usage of **offset** follows the form

offset elem **normal x y z**

where **normal** is the amount to offset the surface along the normal direction, and **x**,**y**,**z** are the translations to be applied. The default normal direction is given by the cross product of the local u and v directions on the VDA surface, taken in that order. **normal** can be negative.

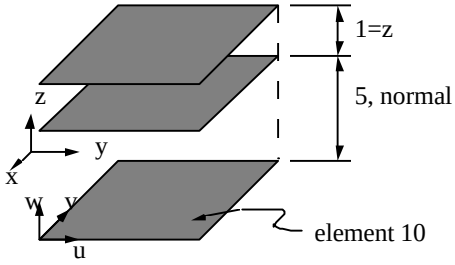
Offset version 2:

Frequently, it is convenient to create a new alias **name** by offsetting and translating an existing **name**. The keyword **goffset** provides this funtion:

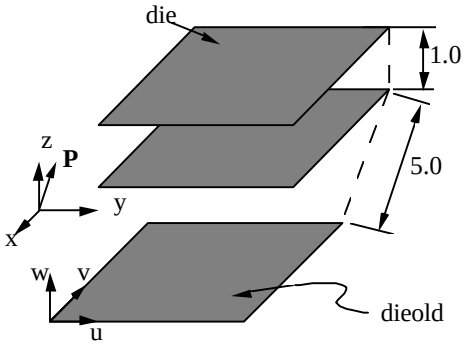
goffset alias **name** $\mathbf{x_c}$ $\mathbf{y_c}$ $\mathbf{z_c}$ **normal x y z** { previous alias **name** }

where **normal**, **x**, **y**, and **z** are defined as in the offset keyword. A reference point $\mathbf{x_c}$, $\mathbf{y_c}$, and $\mathbf{z_c}$ defines a point in space which determines the normal direction to the VDA surface, which is a vector from the origin to $P(x_c,y_c,z_c)$. See example below.

offset 10 5.0 0 0 1.0



offset alias die 1.0 2.0 1.0 5.0 0.0 1.0
{ previous alias dieold }



Finally, several parameters affecting the VDA surface iteration routines can be reset in the file **vda**. These parameters, and their default values in square brackets [ ], are:

gap [5.0] The maximum allowable surface gap to be filled in during the iterations. Points following the surface will effectively extend the edges of surfaces if necessary to keep them from falling through cracks in the surface smaller than this. This number should be set as small as possible while still allowing correct results. In particular, if your VDA surfaces are well formed (having no gaps), this parameter can be set to 0.0. The default value is 5.0.

t**rack** [2.0] A point must be within this distance of contact to be continually tracked. When a point not being tracked comes close to a surface, a global search is performed to find the near surface point. While a point is being tracked, iterations are performed every cycle. These iterations are much faster, but if the point is far away it is faster to occasionally do the global search. The default value is 2.0.

t**rack2** [5.0] Every VDA surface is surrounded by a bounding box. When a global search needs to be performed but the distance from a point to this box is > t**rack2**, the actual global search is not performed. This will require another global search to be performed sooner than if the actual distance to the surface were known, but also allows many global searches to be skipped. The default value is 5.0.

ntrack [4] The number of VDA surfaces for which each point maintains actual distance information. A global lower bound on distance is maintained for all remaining surfaces. Whenever the point moves far enough to violate this global lower

bound, all VDA surfaces must have the global search performed for them. Hence, this parameter should be set to the maximum number of surfaces that any point can be expected to be near at one time (the largest number of surfaces that come together at one point). Setting **ntrack** higher will require more memory but result in faster execution. If **ntrack** is too low, performance may be unacceptably slow. The default value is 4.0.

converge [.01] When surface iterations are performed to locate the near point, iteration is continued until convergence is detected to within this distance (all VDA coordinates are in mm). The default value is 0.01.

iterate [8] Maximum number of surface iterations allowed. Since points being tracked are checked every cycle, if convergence fails it will be tried again next cycle, so setting this parameter high does not necessarily help much. On the other hand, a point converging to a crease in the VDA surface (a crease between patches with discontinuous derivative, for example) may bounce back and forth between patches up to this many times, without actually moving. Hence, this value should not be too large. The default value is 8.

Example for file V=**vda**. It contains the following data:

```
file vda1 vda1.bin {
        alias die {
                sur0001
                sur0003
                offset fce0006 1.5 0 0 120
        }
        alias holder1 { sur008 }
}
file vda2 vda2.bin {
        alias holder2 { sur003 }
}
alias holder { holder1 holder2 }
ntrack 6
gap 0.5
end
```

Explanation:

vda1	This file contains the sufaces/face elements sur0001,sur0003, fce0006, and sur0008.
alias die face	Combines the surface/face elements sur0001, sur0003, and the offsetted surface fce0006 to a global surface.
alias holder1	Defines the surface/face element sur0008 as holder1.
vda2	This file contains the surface/face element sur0003.
alias holder2	Defines the surface/face element sur0003 as holder2.
alias holder	Combines the surfaces holder1 and holder2 into a combined surface holder.
ntrack 6	For each point the actual distances to 6 VDA surfaces are maintained.
gap 0.5	Surface gaps of 0.5mm or less are filled.
end	Closes reading of this file.

APPENDIX J: LS-TAURUS USER'S MANUAL

To open the LS-TAURUS User's Manual, select **LS-TAURUS** in the Bookmark List which should be located at the left-hand side of this window.

LS-DYNA3D

TIME COMPARISONS ON PC AND WORKSTATIONS

RESULTS FROM VERSION 936

(CPU time in seconds - element cycle time in microseconds - time normalized to HP 735)

			PC (20*) 486DX4 100	PC (32) P5 90	PC (16) P5 120	PC (32) P5 166	PC (64) P6 200	IBM (128) RS/6000 550	HP (32) 715 Mod33	HP (128) 735 Mod99
Bar impacting a rigid wall			1,160	476	342	278	126	342	748	186
nodes	1,369									
solids	972	cycle time	217	89	64	52	24	78	140	35
shells	0									
cycles	5,500	normalized	6.2	2.5	1.8	1.5	0.68	2.2	4.0	1.0
Impact of a cylinder into a rail			27,560	11,960	8,420	6,750	2,950	9,860	17,920	4,657
nodes	5,128									
solids	3,667	cycle time	208	90	63	51	22	74	135	35
shells	0									
cycles	36,200	normalized	5.9	2.5	1.8	1.5	0.63	2.1	3.9	1.0
Square plate impacted by a rod			14,590	6,030	4,450	3,590	1,690	4,300	9,450	2,338
nodes	6,856									
solids	1,350	cycle time	194	80	59	48	22	57	125	31
shells	4,824									
cycles	12,200	normalized	6.3	2.5	1.9	1.5	0.71	1.8	4.0	1.0
Box beam buckling			47,880	17,780	14,530	11,430	5,120	14,390	30,030	7,734
nodes	1,911									
solids	0	cycle time	380	141	115	91	41	114	237	61
shells	1,800									
cycles	70,000	normalized	6.2	2.5	1.9	1.5	0.67	1.9	3.9	1.0
Trim saw drop (Black & Decker)			168,510	76,740	54,590	44,240	20,900 [23,140]†	48,330	111,070	29,250
nodes	12,381									
solids	1,140	cycle time	320	146	103	84	40 [44]†	92	211	55
shells	10,726									
cycles	44,420	normalized	5.8	2.6	1.9	1.5	0.73 [0.8]†	1.7	3.8	1.0
Dual airbag-dummy crash			127,060	54,780	43,500 [37,510]†	34,650	16,160 [21,280]†	37,490	74,810	21,005
nodes	25,502									
solids	5,594	cycle time	277	119	95 [82]†	75	35 [46]†	82	163	46
shells	17,757									
cycles	19,670	normalized	6.0	2.6	2.1 [1.8]†	1.6	0.76 [1.0]†	1.8	3.5	1.0

* Megabyte of RAM

† 32 Megabyte of RAM